

**DAHLGREN DIVISION  
NAVAL SURFACE WARFARE CENTER**

Dahlgren, Virginia 22448-5100



**NSWCDD/TR-98/147**

**ZEUS<sup>++</sup>—A GUI-BASED FLOWFIELD ANALYSIS  
TOOL, VERSION 1.0, USER'S MANUAL**

**BY DR. DAVID F. ROBINSON  
WEAPONS SYSTEMS DEPARTMENT**

**FEBRUARY 1999**

Approved for public release; distribution is unlimited.

DESTRUCTION NOTICE—For unclassified, limited distribution documents, destroy by any method that will prevent disclosure of contents or reconstruction of the document.

**DTIC QUALITY INSPECTED 4**

**20000114 036**

**REPORT DOCUMENTATION PAGE**Form Approved  
OBM No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, search existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

<b>1. AGENCY USE ONLY (Leave blank)</b>		<b>2. REPORT DATE</b> February 1999	<b>3. REPORT TYPE AND DATES COVERED</b>	
<b>4. TITLE AND SUBTITLE</b> Zeus <sup>++</sup> —A GUI-based Flowfield Analysis Tool, Version 1.0, User's Manual			<b>5. FUNDING NUMBERS</b>	
<b>6. AUTHOR(s)</b> Dr. David F. Robinson				
<b>7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)</b> Naval Surface Warfare Center Dahlgren Division (Code G72) 17320 Dahlgren Rd Dahlgren, VA 22448-5100			<b>8. PERFORMING ORGANIZATION REPORT NUMBER</b>  NSWCDD/TR-98/147	
<b>9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)</b>			<b>10. SPONSORING/MONITORING AGENCY REPORT NUMBER</b>	
<b>11. SUPPLEMENTARY NOTES</b>				
<b>12a. DISTRIBUTION/AVAILABILITY STATEMENT</b>  Approved for public release; distribution is unlimited.			<b>12b. DISTRIBUTION CODE</b>	
<b>13. ABSTRACT (Maximum 200 words)</b> <p>A User's Manual for the Zeus<sup>++</sup> Flowfield Analysis tool is provided along with a tutorial for setting up and executing a missile geometry with control surfaces. The tool solves the three-dimensional Euler equations to calculate the inviscid flowfield about the geometry. This inviscid solution is then supplied as input to an integral boundary layer solver which calculates the effects of viscosity on the body. Finally, if a more in-depth solution is desired, Zeus<sup>++</sup> can be used to generate a computational domain for use in a full Navier-Stokes solver.</p> <p>By applying the Zeus<sup>++</sup> tool, computational analyses of complex three-dimensional missile geometries can be performed on a personal computer in a matter of minutes. In addition, a computational grid suitable for a full Navier-Stokes solver can be generated in approximately 10 min. This represents significant savings in both manpower and computational costs typically associated with computational fluid dynamics.</p>				
<b>14. SUBJECT TERMS</b>  Zeus <sup>++</sup> computational fluid dynamics (CFD) GUI-based Flowfield			<b>15. NUMBER OF PAGES</b> 96	
			<b>16. PRICE CODE</b>	
<b>17. SECURITY CLASSIFICATION OF REPORT</b> UNCLASSIFIED	<b>18. SECURITY CLASSIFICATION OF THIS PAGE</b> UNCLASSIFIED	<b>19. SECURITY CLASSIFICATION OF ABSTRACT</b> UNCLASSIFIED	<b>20. LIMITATION OF ABSTRACT</b> SAR	

## FOREWORD

The current work is in response to the demand for a computational tool that can be used for rapid and inexpensive aerodynamic analysis of missile-type geometries. Although it is possible to evaluate missile geometries using a full Navier-Stokes computational fluid dynamics procedure (CFD), both budgetary and time constraints often preclude this type of in-depth analysis. The Zeus<sup>++</sup> Flowfield Analysis Tool was developed in order to minimize the cost and turnaround time associated with aerodynamic analysis of missile geometries. The tool can be used to either perform a flowfield analysis, or alternatively, to generate computational domains for a full Navier-Stokes solver.

Initial funding for the work was provided by the Modeling and Simulation Task of the Air and Surface Weaponry Technology Program funded and managed by the Office of Naval Research, Code 351. Additional funding was provided by George Long through the Standard Missile Program, PMS422. The author would like to express his appreciation to both Gil Graff and George Long for their support during this project.

This report has been reviewed by Mr. Thomas A. DelGuidice, Head, Weapons Integration and Technology Branch; and Mr. Bob Steigler, Head, Combat Systems Safety and Engineering Division.

Approved by:

JOHNNY WALTERS, Deputy Head  
Weapons Systems Department

## CONTENTS

<u>Section</u>	<u>Page</u>
1 INTRODUCTION.....	1
1.1 OVERVIEW OF ZEUS <sup>++</sup> .....	1
1.2 DOCUMENTATION .....	3
1.3 SYSTEM REQUIREMENTS.....	3
1.4 CONTACT THE AUTHOR.....	3
1.5 EXECUTE THE CODE .....	4
2 MAIN WINDOW CONTROLS .....	5
2.1 GEOMETRY SKETCH .....	5
2.2 MENU OPTIONS.....	6
2.3 MISCELLANEOUS OPTIONS.....	7
3 GEOMETRY SETUP/EVALUATION .....	8
3.1 COMPUTATIONAL DOMAIN .....	8
3.2 MISSILE GEOMETRY.....	10
3.3 CONTROL SURFACE GEOMETRY .....	16
3.4 GRID GENERATION.....	24
4 INPUT PARAMETERS.....	30
4.1 AERODYNAMIC PARAMETERS .....	30
4.2 INTEGRATION CONTROLS.....	33
4.3 SEPARATION PARAMETERS.....	36
4.4 BOUNDARY LAYER CONTROLS.....	38
4.5 OUTPUT CONTROLS.....	43
5 CODE EXECUTION .....	45
5.1 CONSOLE OUTPUT.....	46
5.2 ERROR MESSAGES.....	49
6 POST-PROCESSOR.....	51
6.1 ASCII DATA.....	51
6.2 SINGLE RUN PLOTS .....	53
6.3 MULTI-RUN PLOTS .....	54
7 REFERENCES .....	56
DISTRIBUTION .....	(1)

## ILLUSTRATIONS

<u>Section</u>	<u>Page</u>
1 MAIN VIEW SCREEN.....	5
2 PROGRAM CONTROL PARAMETERS.....	7
3 MISSILE GEOMETRY SELECTION.....	10
4 CONIC SECTION PARAMETERS .....	11
5 TANGENT OGIVE PARAMETERS .....	11
6 SECANT OGIVE PARAMETERS.....	12
7 VON KARMAN OGIVE PARAMETERS .....	13
HAACK SERIES NOSE PARAMETERS .....	13
9 POWER SERIES NOSE PARAMETERS .....	14
10 MISCELLANEOUS OPTIONS .....	15
11 MISSILE CROSS-SECTION PARAMETERS.....	15
12 CONTROL SURFACE GEOMETRY SELECTION .....	16
13 CONTROL SURFACE PARAMETERS.....	17
14 SAMPLE 1—USER DEFINED CONTROL SURFACE PARAMETERS .....	19
15 SAMPLE 2—USER DEFINED CONTROL SURFACE PARAMETERS .....	20
16 USER DEFINED CONTROL SURFACE INPUTS.....	20
17 MODIFIED WEDGE CONTROL SURFACE PARAMETERS.....	21
18 NACA 4-DIGIT AIRFOIL PARAMETERS .....	22
19 EULER GRID GENERATION PARAMETERS.....	25
20 NAVIER-STOKES GRID GENERATION PARAMETERS.....	26
21 SAMPLE CROSS-SECTION OF NAVIER-STOKES GRID .....	27
22 NAVIER-STOKES BOUNDARY CONDITION PARAMETERS .....	28
23 AERODYNAMIC SWEEP PARAMETERS.....	31
24 GENERAL AERODYNAMIC INPUT PARAMETERS .....	32
25 NUMERICAL INTEGRATION CONTROLS .....	34
26 SEPARATION MODELING PARAMETERS .....	37
27 BOUNDARY LAYER EXECUTION PARAMETERS .....	39
28 BOUNDARY LAYER MODELING PARAMETERS.....	40
29 BOUNDARY LAYER OUTPUT PARAMETERS.....	41
30 OUTPUT CONTROL PARAMETERS.....	43
31 EXECUTIVE CONTROL PARAMETERS.....	46
32 EULER CONSOLE OUTPUT.....	47
33 BOUNDARY LAYER CONSOLE OUTPUT .....	48
34 EULER GRID REFERENCE DIAGRAM.....	52
35 CONTOUR PLOT CONTROLS .....	54
36 24 PLOT CONTROLS.....	55

## CHAPTER 1: INTRODUCTION

### 1.1 OVERVIEW OF ZEUS<sup>++</sup>

In the past decade, significant advances have been made in the field of computational fluid dynamics (CFD). These include improvements in computational speed, numerical algorithm efficiency, and modeling (turbulence and combustion). Because of this progress, it is now commonplace to see accurate CFD solutions to very complex, large scale, three-dimensional flowfields. It should be noted, however, that a number of limitations remain. First, existing CFD solvers are quite complex and extensive training is required prior to generating a solution. The user must be trained not only in basic aerodynamics, but also in computational domain development and numerical methods. Second, the computational costs are still extensive. Complex cases can still take days, weeks, or even months of expensive workstation or supercomputer time. Because of these limitations, parametric type design studies are often infeasible, and therefore, the use of CFD as an engineering design/development tool has been limited.

The goal of the current work is to significantly reduce both the costs and turnaround time associated with CFD solutions of missile type geometries with arbitrary control surfaces. The current work began with the ZEUS flowfield solver<sup>2,4,7,8</sup>. The ZEUS code is a supersonic space marching Euler solver capable of computing both internal and external flowfields. The current work involved modifying the ZEUS code to make it more amenable to solving for the forces/moments on missile type geometries with arbitrary control surfaces. A number of limitations in the original code motivated the current work:

- 1) For each geometry under consideration, the user must develop a Fortran source code that provides a geometric description of the missile surface. In addition, code that specifies the control surfaces and switching logic for the boundary conditions must be generated.
- 2) The user must be familiar with a number of separate codes in order to implement a ZEUS run.

These include a separate code for each of the following tasks:

- a) Cone/Blunt - Generate an initial profile by running one of the two codes.
- b) Convert - Convert the initial profile to a form readable by ZEUS
- c) ZEUS - Solves the Euler form of the equations of motion.
- d) BL - Performs a boundary layer analysis of the inviscid flowfield.

Since the user must understand all of the inputs/outputs from each of the above codes, this significantly increases the amount of training required.

- 3) The setup time required before executing the code can be significant (typically on the order of hours/days). In addition, if parametric studies are desired, a separate set of input files must be generated for each case under consideration. This is both a time consuming and error prone process when a large number of cases are desired.
- 4) Inherent limitations (space marching) in the ZEUS code prevent the solution of any flowfield with subsonic regions.

The modifications made to the ZEUS code focused on the following four primary areas of improvement:

- 1) Minimize the required training - This was done by developing a graphical user interface, modifying the grid generation capabilities, and developing simple to use pre- and post-processing features. The GUI allows for most of the complexity of the code to be hidden from the user. Instead of developing Fortran modules to specify the missile and control surface geometries, the current tool (Zeus++) uses a series of point-and-click GUI menus to setup the required geometry files. All of the complexities of generating a three-dimensional grid and blending control surfaces to the missile have been hidden from the user. The GUI interface also significantly simplifies the post-processing of the results.
- 2) Reduce the setup/execution time - The interface reduces the setup time required to a few minutes vs. many hours/days for the original ZEUS code. In addition, the user can setup the code to sweep through a variety of parameters (Mach number, angle of attack, angle of yaw, and roll angle) by simply specifying an initial, final, and incremental value for each of the parameters. All of the complexity involved with generating numerous input files has been hidden from the user. This feature allows for parametric type studies with no additional setup time required.
- 3) Minimize the costs - The costs associated with computational fluid dynamic solutions are often prohibitively high, and therefore, prevent the application of CFD during the design/analysis phase of missile development. The two primary influences on these costs are personal and computational requirements. The personal costs are inflated because you must have someone with extensive training performing a very time consuming task. Zeus++ reduces the personnel costs by minimizing the required training and reducing the time required to setup a run. The second factor, computational cost, is high because the ZEUS code was developed for use on expensive Unix workstations. These costs were reduced by porting the ZEUS code to less expensive personal computers and by developing a GUI that runs under a widely available operating system (Windows NT and Windows 95).
- 4) Extend the range of applications for the code - There are a number of cases where either the ZEUS code cannot be applied, or where the results may be suspect due to the approximate nature of the solution procedure (space marching Euler). For these cases, the Zeus++ tool includes a grid generator which will

output a three-dimensional grid and the appropriate boundary conditions which can then be applied as input for the GASP<sup>1</sup> full Navier-Stokes flowfield solver. It should be noted that generation of complex three-dimensional missile grids with control surfaces often took days to develop. The Zeus<sup>++</sup> tool reduces this to a matter of minutes on a personal computer.

## 1.2 DOCUMENTATION

The documentation for the Zeus<sup>++</sup> tool is presented in one of two forms: online help or a printed user's manual. It should be noted that the data contained within the two forms are identical.

To access the online help press the 'F1' key during execution of the Zeus<sup>++</sup> code. If the user does not understand one of the requested inputs, pressing the 'F1' key will display the help file for that specific topic. Alternatively, the user can search through the help documentation online by clicking on 'Help' from the main view screen.

A hard copy of the Zeus<sup>++</sup> user's manual can be obtained by executing the code (double clicking on ZeusPP.exe) and selecting "Print User's Manual". Alternatively, the manual may be printed by selecting "Help" and then "Print User's Manual" from the main window.

Note: The manual is provided in Adobe pdf format. The Adobe Acrobat reader may be obtained from Adobe's home page at "<http://www.adobe.com/acrobat>". Alternative forms of the User's manual can be supplied upon request if the pdf format cannot be viewed/printed.

## 1.3 SYSTEM REQUIREMENTS

Processor:	Intel 486 compatible (233 MHz Pentium II or higher recommended)
Operating System:	Microsoft Windows NT 4.0 (recommended), Windows 95, or Windows 98
Plotting Package:	Tecplot v7.0 (Optional)
Hard Disk Space:	Approximately 3 Megabytes for the Zeus <sup>++</sup> code and associated files. The size of the output data generated by Zeus <sup>++</sup> is highly case dependent (grid resolution, # of sweeps, etc...) and can range from a few megabytes to a few hundred megabytes.

## 1.4 CONTACT THE AUTHOR

E-mail requests for a copy of the code to the author at the following:

Dr. David F. Robinson (Code G72)  
Naval Surface Warfare Center  
17320 Dahlgren Rd.  
Dahlgren, VA 22448-5100  
[robinsondf@nswc.navy.mil](mailto:robinsondf@nswc.navy.mil)

Voice: (540) 653-3954

Fax: (540) 653-8268

## 1.5 EXECUTE THE CODE

Execution of the code begins by double clicking on the program 'ZeusPP.exe', entering a name for the current case, and then clicking on 'start New Case'.

Note: 1) The name entered cannot contain blank spaces.

2) A tutorial for setting up and executing a new case is provided in Appendix A.

3) The code must be executed from the main ZeusPP directory. To execute one of the sample cases provided, copy the file from the 'samples' directory to the ZeusPP main directory.

To open an existing case (\*.zpp) click on 'ZeusPP.exe' and select 'Open Existing Case'. From the main menu click on 'File/Open' and select the desired case. Alternatively, simply double click on the file ('filename.zpp').

## CHAPTER 2: MAIN WINDOW CONTROLS

### 2.1 GEOMETRY SKETCH

As the geometry of the missile and its control surfaces are specified, two-dimensional sketches of the side and frontal views are displayed in the main view screen (see example below). There are a variety of methods for manipulating the sketch, each of which is listed in the upper left corner of the main view screen and discussed in detail below.

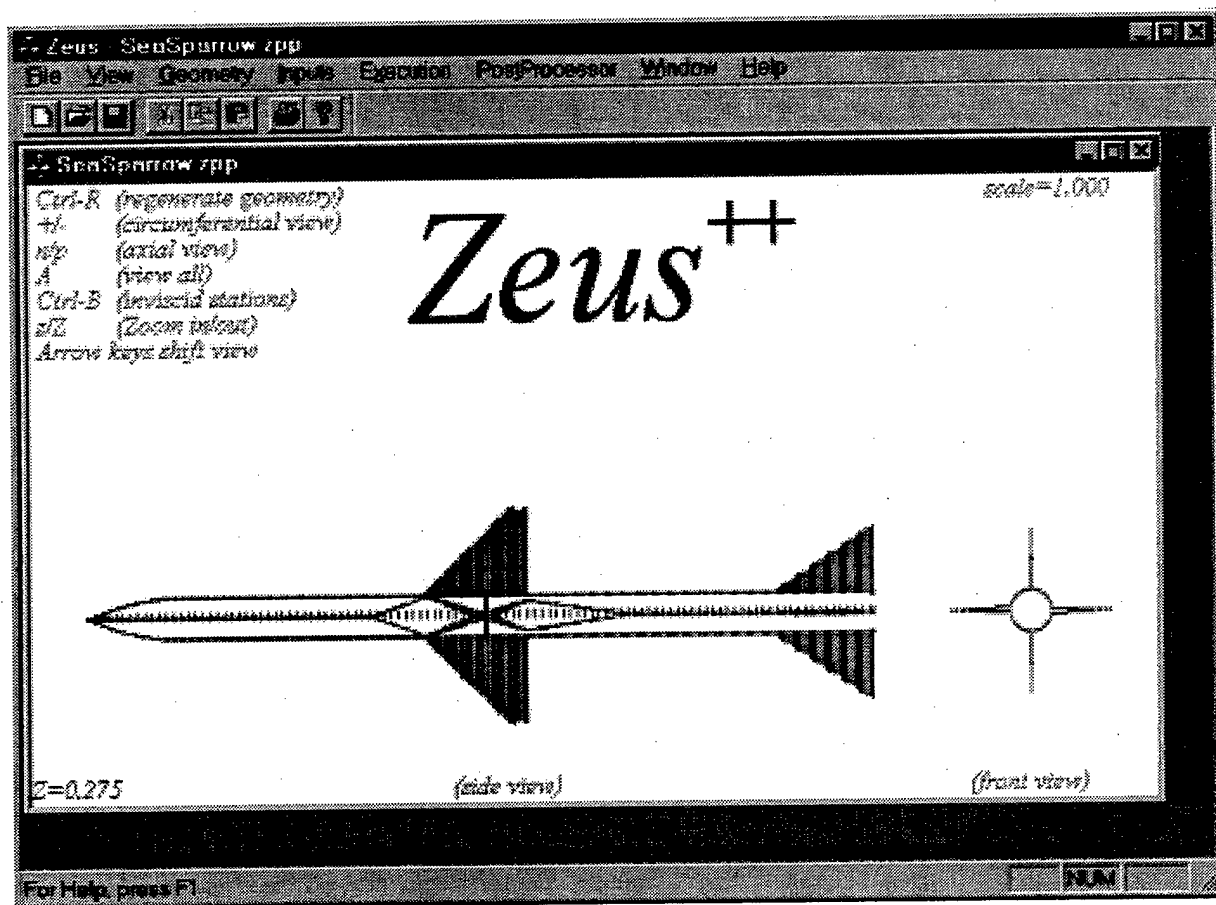


FIGURE 1. MAIN VIEW SCREEN

- **Ctrl-R** - Pressing the 'Control' and 'R' keys simultaneously will regenerate the geometry display shown in the main window. This should be done any time the missile geometry is modified and an updated view is desired. Note that this view is for display purposes only, and therefore, to limit computational effort, a low-resolution representation of the surface geometry is provided. This is not representative of the computational domain employed during the solution

procedure. If desired, the resolution of the display can be increased/decreased by modifying the number of points in the View/Options section.

- **+/-** - Pressing the '+' and '-' keys steps the views circumferentially through each of the zone edges. '+' shows the next and '-' shows the previous circumferential view.
- **n/p** - Pressing the 'n' and 'p' keys steps the front view axially down the missile. 'n' and 'p' go to the next and previous axial location, respectively. Pressing 'N' or 'P' has the same effect, however, larger axial steps are taken. The current axial location is shown in the lower left corner of the main view screen ( $z=0.275$ ).
- **A/a** - Pressing the 'A' or 'a' key redraws all of the axial stations and circumferential planes.
- **Ctrl-B** - Pressing the 'Control' and 'B' keys simultaneously will display the axial stations where the inviscid grid was output. These stations are shown as blue tic marks along the centerline of the missile geometry. *This feature cannot be employed until after an inviscid run has been completed.*

Note: The viscous boundary layer solver requires the inviscid Euler solution as a boundary condition. Therefore, the inviscid solution must be performed before running the viscous code. To obtain an accurate viscous approximation an adequate number of inviscid profiles must be provided such that all relevant geometric features are spatially resolved. The inviscid profiles are written out at the user-specified intervals provided in the boundary layer inputs section.

- **'z'/'Z'** - Pressing the 'z' or 'Z' keys zooms in/out on the main view screen. The amount of scaling applied to the original sketch is shown in the upper right hand corner. To return to the original settings, press Ctrl-R to regenerate the sketch.
- **Arrow keys** - The arrow keys are used to translate the sketches in the main window.

## 2.2 MENU OPTIONS

<u>File</u>	Controls file I/O and printing.
<u>View</u>	Toolbar views and runtime options.
<u>Geometry Setup</u>	Define the missile and control surface geometries as well as the computational domain.
<u>Inputs</u>	Input free-stream conditions, runtime parameters, reference conditions, ...etc.
<u>Execution</u>	Executing the code
<u>Post-Processor</u>	Post-Processing
<u>Window</u>	Layout and orientation of windows
<u>Help</u>	Help controls and 'About' information (author, version #, and contact information)

## 2.3 MISCELLANEOUS OPTIONS

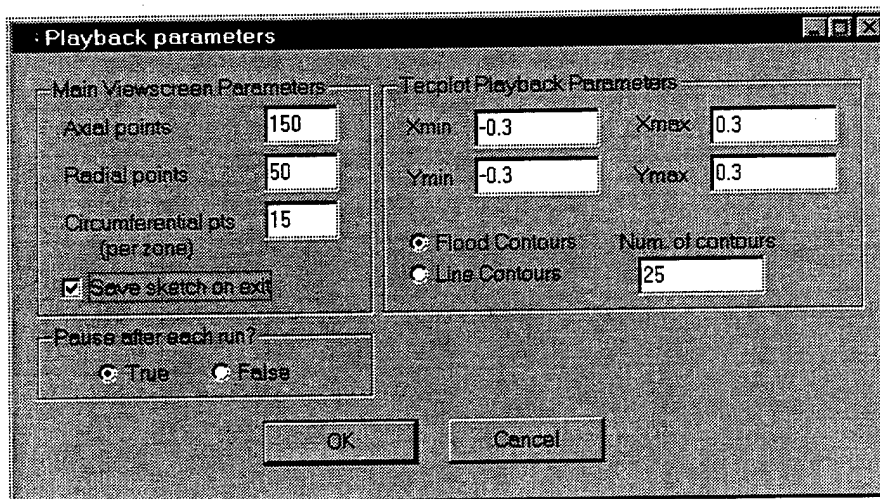


FIGURE 2. PROGRAM CONTROL PARAMETERS

- **Number of Points** - The sketch shown in the main window is a low-resolution representation of the current missile geometry. To increase/decrease the resolution of the geometric display simply increase/decrease the appropriate points parameter (axial, radial, or circumferential). Increasing the resolution will increase both the time to display the sketch as well as the size of the output data file (\*.zpp).
- **Save sketch** - The output data file for the Zeus<sup>++</sup> code (\*.zpp) contains all of the input parameters as well as the coordinates for the geometry sketch shown in the main window. To minimize the size of the output files (\*.zpp) de-select the 'save sketch on exit' dialog box shown above. This will significantly decrease the size of the output file (approximately 3k vs. 1-2 meg.); however, when the case is restarted, the geometry sketch must be regenerated. If the size of the output file is not important then it is recommend that this option be selected to minimize the computational effort required at startup.
- **Tecplot Parameters** - A number of Tecplot macro files (contour plots, movie files, ...etc.) are generated by the Zeus<sup>++</sup> code in order to simplify post-processing. The following parameters, used in the macro files, are specified in the View/Options section.
 

$X_{min}$ , $X_{max}$	- The minimum/maximum x-axis range for the movie files.
$Y_{min}$ , $Y_{max}$	- The minimum/maximum y-axis range for the movie files.
Flood/Line	- Specify either flood or line drawing for the contour plots.
# of contours	- Specify the number of contour lines to be drawn.
- **Pause** - Zeus<sup>++</sup> runs a number of external programs from the command prompt. If the 'Pause after each run' feature is selected then the command prompt will remain active (i.e., paused) and prompt the user to 'press any key' before exiting. If the pause feature is false, the Zeus<sup>++</sup> code will automatically close the command prompt upon termination of the code.

## CHAPTER 3: GEOMETRY SETUP/EVALUATION

### MENU OPTIONS

<u>Grid Parameters</u>	Specify the grid parameters for both the Euler and Navier-Stokes grids. (# of grid points, clustering, # of zones, and location of outer boundary).
<u>Missile Sections</u>	Define the missile geometry
<u>Control Surfaces</u>	Define the control surface geometry
<u>Grid Generation</u>	Generate the Euler and Navier-Stokes computational domains.

### 3.1 COMPUTATIONAL DOMAIN

#### 3.1.1 Grid Parameters

Recall the Zeus<sup>++</sup> tool has two primary functions. The code serves not only as an approximate flow solver, but also as a Navier-Stokes grid generator. Therefore, two separate sets of grid parameters are specified when setting up the computational domain. In the first, the user inputs the parameters for the computational domain that will be used during the Euler and boundary layer solutions of the flowfield. In the second, the input parameters are used to generate the Navier-Stokes grid and appropriate boundary conditions.

#### Mesh Dimensions

- *Number of zones* - The number of zones is equal to the maximum number of control surfaces at a given axial location (i.e., if you have 4 canards, 3 dorsal fins, and 5 tails fins then you must enter '5' as the number of zones).
- *No control surfaces* - Select this box if you are considering a case with no control surfaces.
- *Number of radial cells* - The number of cells located radially between the body surface and the outer boundary of the computational domain.
- *Default number of circumferential cells/zone* - Changing this parameter will change the number of circumferential cells/zone for all zones.
- *Number of cells/zone* - The number of cells/zone refers to the circumferential direction (i.e., between zone edges) and does not have to be the same for all zones. Change an individual zone by clicking on the desired zone and entering a new value.

## Mesh Clustering

The flowfield solver typically does not require mesh clustering because there is no boundary layer to resolve. However, in certain cases, mesh clustering (radial and circumferential) can be used to more accurately capture specific flowfield features (vortices, shocks, etc). Note that mesh clustering is required for the Navier-Stokes grid in order to resolve the boundary layer.

- *Radial clustering* - Specify the grid clustering between the body surface and the outer boundary. A value of 1.0 provides equal spacing and a value of 0.0 provides maximum clustering at the surface.

Note: 1) Two sets of radial clustering are required. These correspond to the radial clustering at the initial and final (nose and tail) axial stations. This provides the ability to have more radial clustering at the nose than at the tail of the missile, or vice-versa. The radial clustering parameter is interpolated linearly between the two values as you progress axially down the missile.

2) Clicking the 'Boundary Layer Thickness' button provides an estimate of the boundary thickness (laminar and turbulent) at a given axial station. The values are calculated using flat plate boundary layer assumptions and can be used to estimate the amount of radial clustering required for a viscous grid. This calculation requires a unit Reynolds number and an axial location. Initial spacing for the viscous grid is typically taken as 1/30th of the estimated boundary layer thickness.

- *Circumferential clustering* - Specify the clustering at the zone edges. A value of 1.0 provides equal spacing and a value of 0.0 provides maximum clustering at the zone edges (i.e., circumferentially between control surfaces).

## Outer Boundary

- *Distance to outer boundary* - If the outer boundary of the computational domain is not tracking a shock (i.e., Free-Stream starting condition) then the radial distance from the body surface to the outer boundary is specified in terms of body radii (typically 10-20 body radii for an inviscid solution).
- *Growth of outer boundary* - In certain cases it is desirable for the radial distance between the body surface and the outer boundary to increase as you march axially down the missile. This parameter specifies the angle (in degrees) which the outer boundary grows relative to the body surface.

Note: The above parameters are used during the Navier-Stokes grid generation. They are also used during the Euler solution if the outer boundary does not track the conical/bow shock. The type of outer boundary applied by the flowfield solver is specified in the Integration Parameters section.

### Copy Parameters

- *Select 'Copy Parameters' button to copy the grid parameters from the alternate grid (i.e., from the Navier-Stokes grid if you are in the Euler grid setup, or vice-versa)*

## 3.2 MISSILE GEOMETRY

### 3.2.1 Select Geometry Type

The first step in setting up the geometry is to define all of the sections that comprise the missile surface (i.e., not control surfaces). First, select the desired type of nose and then click on 'Add New Section'. You will be prompted for the geometric parameters used to generate the desired type of nose section. After the nose section has been specified, repeat the above process for any remaining sections (i.e., afterbody, boat-tail, etc...). Note that the geometry sections must be entered in the order in which they occur on the geometry.

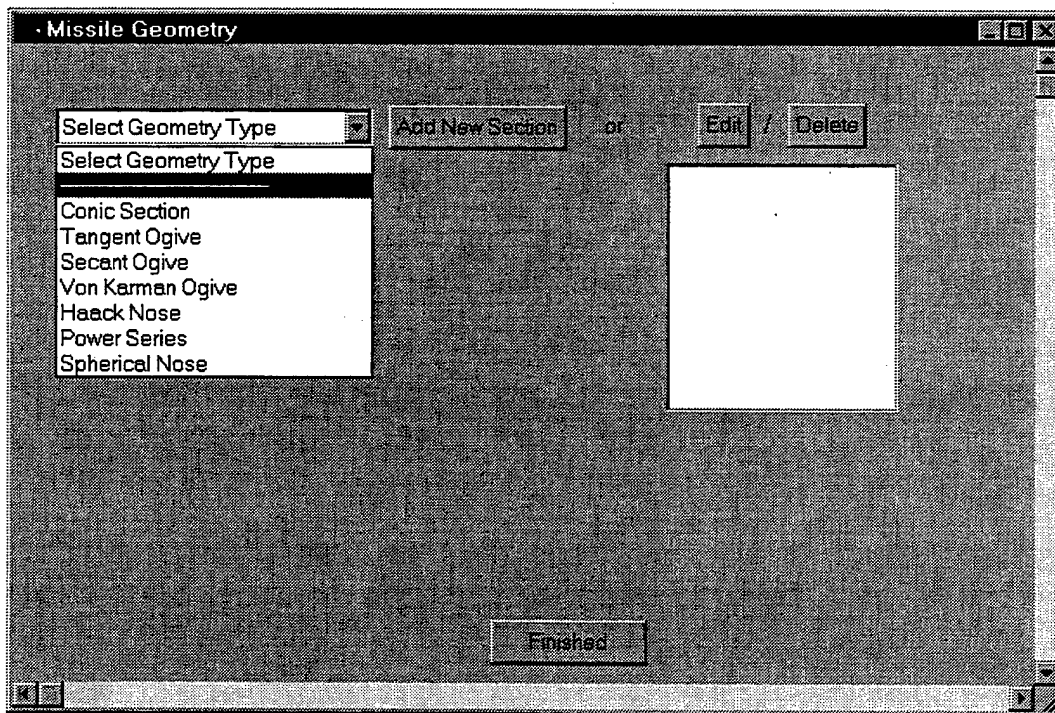


FIGURE 3. MISSILE GEOMETRY SELECTION

It should be noted from the above figure, that there are a variety of possible missile sections. A sample sketch, a description of the required parameters, and the generating equations are shown below for each of the available geometry sections. In all of the geometric formulas discussed below, 'z' is the axial distance from the start of the given section.

### 3.2.2 Conic



Conic Section

FIGURE 4. CONIC SECTION PARAMETERS

A variety of geometries can be described using the conic input section. These include those shown in the sketch above, namely, a nose cone, a cylindrical section, and a boat-tail flare. The parameters required to generate a conic section are shown in the above figure.

### 3.2.3 Tangent Ogive



Tangent Ogive

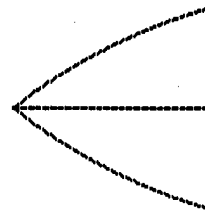
FIGURE 5. TANGENT OGIVE PARAMETERS

A sample sketch of a tangent ogive is shown above along with the parameters required to generate the section. Note from the sketch that the missile section is tangent to the horizontal at the aft axial station, hence the name, tangent ogive. The geometry is generated using the following formula<sup>10</sup> along with the user specified parameters R and L, where R is the base radius and L is the length of the section.

$$r = \sqrt{\rho^2 + (z - L)^2} + (R - \rho)$$

$$\rho = \frac{R^2 + L^2}{2R}$$

### 3.2.4 Secant Ogive



Secant Ogive

FIGURE 6. SECANT OGIVE PARAMETERS

A sample sketch of a secant ogive is shown above along with the parameters required to generate the section. The geometry is generated using the following formula<sup>10</sup> along with the user specified parameters  $L$ ,  $R$ , and  $\rho$ , where  $L$  is the length of the section,  $R$  is the base radius, and  $\rho$  is the radius of curvature.

$$r = \sqrt{\rho^2 + (z - h)^2} + k$$

$$h = \rho \cos \alpha$$

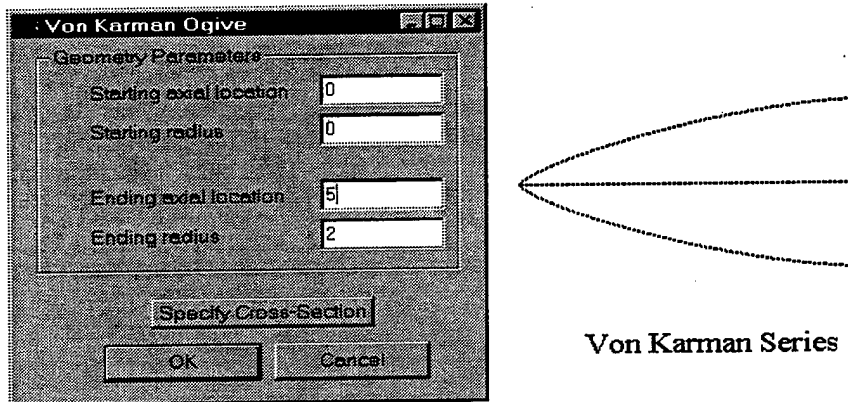
$$k = \rho \sin \alpha$$

$$\alpha = \gamma - \theta$$

$$\theta = \tan^{-1} \left( \frac{R}{L} \right)$$

$$\gamma = \cos^{-1} \left( \frac{\sqrt{L^2 + R^2}}{2\rho} \right)$$

### 3.2.5 Von Karman Ogive



Von Karman Series

FIGURE 7. VON KARMAN OGIVE PARAMETERS

A sample sketch of a Von Karman ogive is shown above along with the parameters required to generate the section. The geometry is generated using the following formula<sup>10</sup> along with the user specified parameters R and L, where R is the base radius and L is the length of the section.

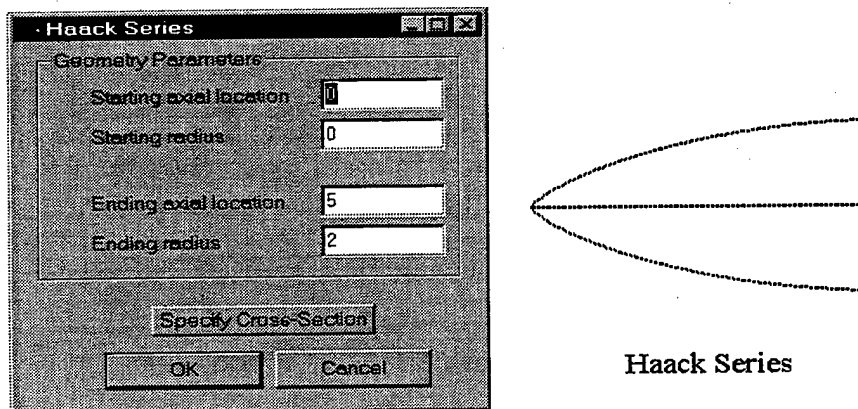
$$\bar{r} = \sqrt{\frac{\left( \phi - \frac{\sin(2\phi)}{2} + C \sin^3 \phi \right)}{\pi}}$$

$$\phi = \cos^{-1}(1 - 2\bar{z})$$

$$C = 0$$

$$\bar{r} = \frac{r}{R}, \quad \bar{z} = \frac{z}{L}$$

### 3.2.6 Haack Series



Haack Series

FIGURE 8. HAACK SERIES NOSE PARAMETERS

A sample sketch of a Haack series is shown above along with the required input parameters. The geometry is generated using the following formula<sup>10</sup> along with the user specified parameters  $R$  and  $L$ , where  $R$  is the base radius and  $L$  is the length of the section.

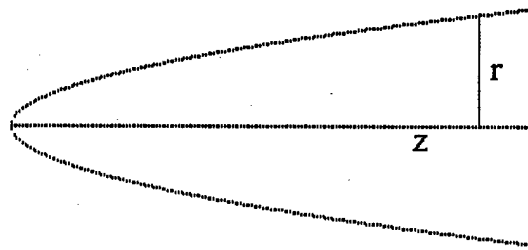
$$\bar{r} = \sqrt{\frac{\left( \phi - \frac{\sin(2\phi)}{2} + C \sin^3 \phi \right)}{\pi}}$$

$$\phi = \cos^{-1}(1 - 2\bar{z})$$

$$C = \frac{1}{3}$$

$$\bar{r} = \frac{r}{R}, \quad \bar{z} = \frac{z}{L}$$

### 3.2.7 Power Series



Power Series

FIGURE 9. POWER SERIES NOSE PARAMETERS

A sample sketch of a Power series section is shown above along with the required input parameters. The geometry is generated using the following formula<sup>10</sup> along with the user-specified parameters  $R$ ,  $L$ , and  $n$ , where  $R$  is the base radius,  $L$  is the length of the section, and  $n$  is the power series exponent.

$$r = z^n$$

### 3.2.8 Spherical Nose

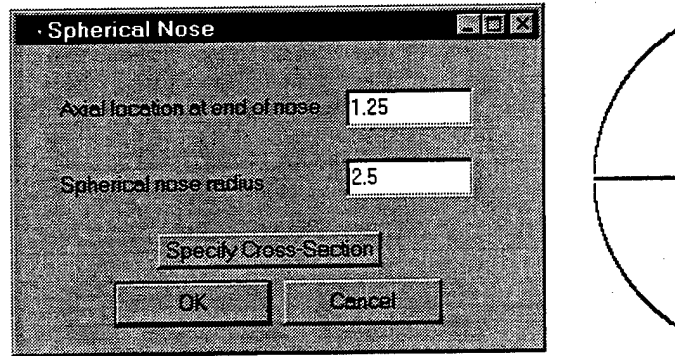


FIGURE 10. SPHERICAL NOSE PARAMETERS

A sample sketch of a spherical nose section is shown above along with the parameters required to generate the section. For the spherical nose, there will be a small region where the flow is subsonic. Because the Euler solver employs a marching procedure, the Zeus code cannot be applied in this subsonic region. Therefore, the blunt body solver must be applied over the entire subsonic region. The outflow plane from the blunt body solver can then be taken as the inflow plane for the Euler solver. The input parameters for the blunt body solver are specified in the Integration Parameters section.

### 3.2.9 Cross-Sectional Geometry

For each missile section defined, a cross-sectional geometry must be specified. If a cross-section is not designated the default value of 'circular' will be used.

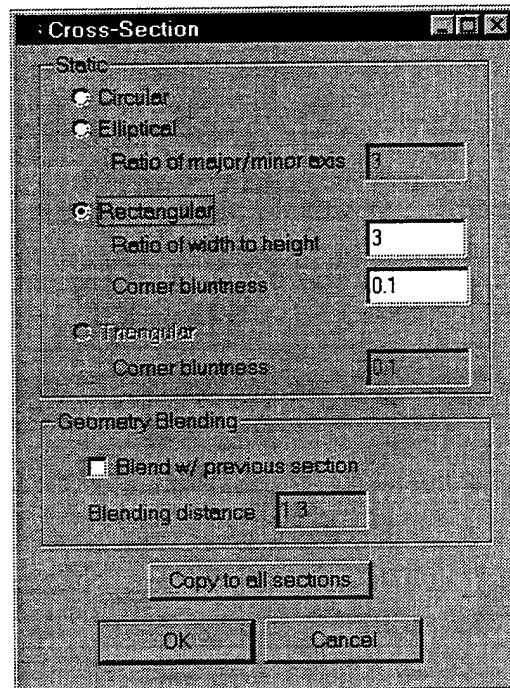


FIGURE 11. MISSILE CROSS-SECTION PARAMETERS

- *Cross-Section* - Select the desired cross-sectional geometry.
- *Corner Bluntness* - Specify a value between zero (no blunting) and 0.999 (completely blunted corners) for the Rectangular cross-sectional geometry.
- *Blending* - It is often desirable to have varying cross-sectional geometries for different sections of the missile (i.e., circular nose cone and elliptical afterbody). For these cases, the cross-sectional geometry cannot be abruptly changed, but rather, must be blended axially between the two sections. If blending is desired, then select the 'Blend w/ previous section' box and specify a blending distance in ft/m.

### 3.3 CONTROL SURFACE GEOMETRY

#### 3.3.1 Select Geometry Type

After the missile geometry has been defined, the next step is to add the control surfaces. If no control surfaces will be examined, then the 'No Control Surfaces' parameter must be selected in the grid parameter section. Select the desired type of control surface from the drop down menu and click on 'Add Control Surface'.

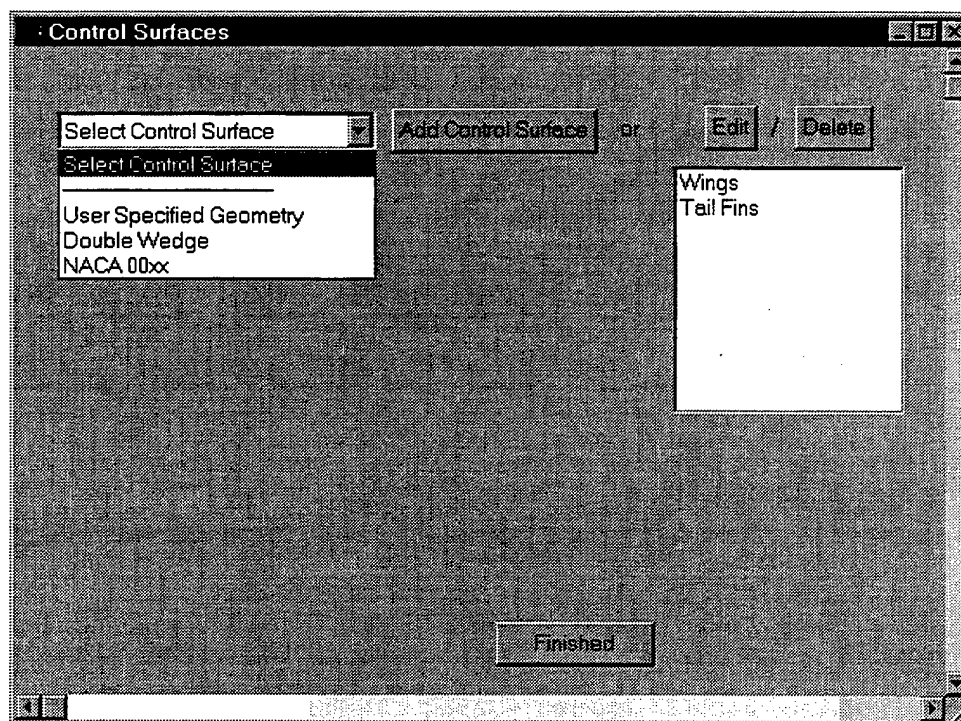


FIGURE 12. CONTROL SURFACE GEOMETRY SELECTION

It should be noted from the above figure, that there are a variety of possible control surfaces. A sample sketch, a description of the required parameters, and the generating equations are shown below for each of the available control surfaces.

### 3.3.2 Control Surface Parameters

Before specifying the geometry of the control surfaces, a number of parameters (see below) which describe the location/orientation of the surfaces on the missile must be specified. These parameters, along with a description of each, are shown below.

**Control Surface Properties**

Enter a label for the control surface: Wings

Enter the axial location of the hinge line (i.e. location of the control surface on the missile): 0.2743

☐ Overlaps previous control surface

**Control Surface Angles (degrees)**

Offset Angles	Deflection Angles	Aileron/Slat Angles
90.0	5	0
180.0	0	0
270.0	10	0
360.0	0	0

(Right Click to delete a control surface)

**Deflect about**

☐ Model Slats/Ailerons

☒ Fore of fin (Slat)

☐ Deflect after fin (Aileron)

flap: 0

OK Cancel

FIGURE 13. CONTROL SURFACE PARAMETERS

- **Label section** - Enter a text label that describes the control surface.
- **Hinge line** - Enter the axial distance to the control surface hinge line measured from the missile nose. The hinge line is the point on the control surface about which deflections are made.
- **Overlap previous control surface** - Select this option when two different types of control surfaces are desired at a given axial location (i.e., different size vertical and horizontal stabilizers). A detailed description of this feature is provided in the [Linked Fins](#) section.
- **Offset angles** - The offset angles describe the orientation of the fins. For the '+' configuration the corresponding angles are 90°, 180°, 270°, and 360°. Similarly, for the 'x' configuration the angles are 45°, 135°, 225°, and 315°.

Note: 1) The Zeus<sup>++</sup> code does not require any form of symmetry, and therefore, any desired combination of offset and/or deflection angles may be specified.

2) The control surfaces must lie on a zonal boundary. Therefore, the zone orientations are equivalent to the offset angles. In other words, the zone edges will occur at the above-specified offset angles.

- *Deflection angles* - The deflection angles are specified in a clockwise manner. In other words, specifying +10 for both the 90° and 270° fins would pitch the former up and the latter down 10°. If a symmetric deflection is desired, the required inputs are +10 for the 90° and -10 for the 270° fin.

Note: The Zeus<sup>++</sup> code does not require any form of symmetry, and therefore, any desired combination of offset and/or deflection angles may be specified.

- *Aileron/Slat Angles* - In addition to deflecting the entire surface about the hinge line (see above), it is also possible to deflect a portion of a control surface. This feature allows for the modeling of either Aileron or Slat type controls. To model either a slat or an aileron, first select on the 'Model Slats/Ailerons' and then enter the appropriate deflections in the 'Ailerons/Slats Angles' dialog box. Finally, enter the 'ilocation' of either the slat termination point or the aileron initiation point. The 'iloc' parameter corresponds to the index of the termination/initiation point along each plane defining the control surface and is explained in the User Defined Geometry section.

Note: When modeling either flaps or slats, it is not possible to deflect a portion of the trailing/leading edges. The entire edge is deflected about the 'iloc' point by the amount specified in the Aileron/Slat Angles dialog box.

- *Delete a fin* - The number of fins is equal to the maximum number of zones specified in the grid parameter section. However, it is possible to delete a fin by right-mouse clicking on the desired fin. Deletion causes the offset, deflection, and the Aileron/Slat angles to be replaced with '——' to indicate that they have been eliminated. To add the fin back into the calculation, simply right click on it a second time. This feature is useful if a variable number of fins are desired for the different sets of control surfaces (i.e., 2 canards and 4 tail fins).

### 3.3.3 User Defined Geometry

The most general method of specifying the control surface geometry is the 'User Defined Geometry' option. For this method, the actual geometry points (x, y, z) are prescribed by the user. The generality of this method allows nearly any form of symmetric control surface to be examined. Two example cases will be discussed in the current section, the first of which is a simple modified wedge control surface. The figure shown below represents the upper half of the symmetric control surface and would mate to the missile body along edge '1-2-3-4'.

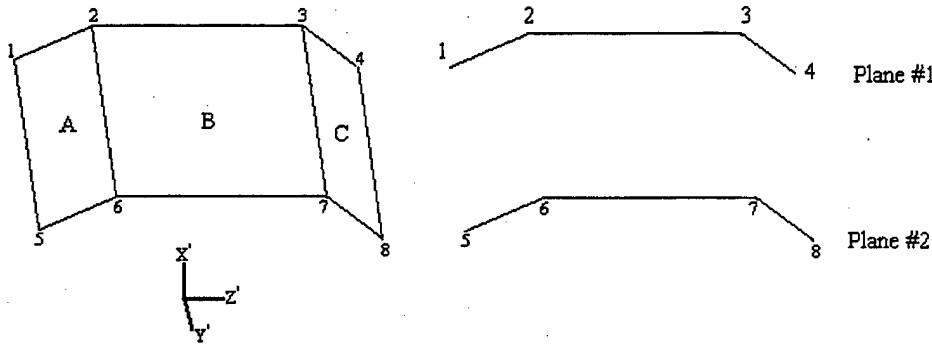


FIGURE 14. SAMPLE 1—USER DEFINED CONTROL SURFACE PARAMETERS

The Zeus<sup>++</sup> tool requires that the control surfaces be described by a series of planar surfaces (see 'A', 'B', and 'C' above). In other words, 'A', 'B', and 'C' each have a single unit outer normal. This allows the code to calculate unit normals to each section, and hence, to generate a computational domain around the control surface. The geometry is specified by decomposition into a series of planes, all of which must have the same number of node points. For the figure shown above the surfaces 'A', 'B', and 'C' are described by specifying two planes with four points on each plane (Points '1', '2', '3', and '4' define Plane #1, Points '5', '6', '7', and '8' define Plane #2). The first plane (Plane #1) is that which corresponds to the root chord (i.e., lies along the missile surface). Successive planes march outwards until the final plane (Plane #2) is reached at the tip chord.

Note: 1) The coordinate system for entering the control surface points is based on a fin local system.

'x' is the thickness of the control surface.

'y' is the radial distance measured relative to the missile surface. (y=0 corresponds to the surface of the missile).

'z' is the axial distance measured relative to the hinge line.

2) The control surfaces must be symmetric, and therefore, only the upper half of the control surface is specified in the 'User Defined Geometry' section. Because of this symmetry requirement, the leading and trailing edges of the fin must have zero thickness (x=0). This requires that x be zero for the first and last points on each plane describing the control surface.

3) Each plane must contain the same number of node points (four for the above example).

The above example can be used to model either slats or flaps. Note that four points are used to define each of the planes (Plane #1 and Plane #2). To model slats, set the 'iloc' parameter to '2', select the 'Fore of iloc (Slats)' dialog, and specify the slat deflection angle in the Control Surface Parameters section. This causes all points ahead of 'iloc' (surface 'A') to be deflected by the specified slat angle. Similarly, to model flaps, set 'iloc' equal to 3 and select the 'Aft of iloc (Flaps)' dialog to deflect all points aft of the third planar point (i.e., deflect surface 'C').

The second example represents a more complex control surface where three planes must be considered to properly define the geometry.

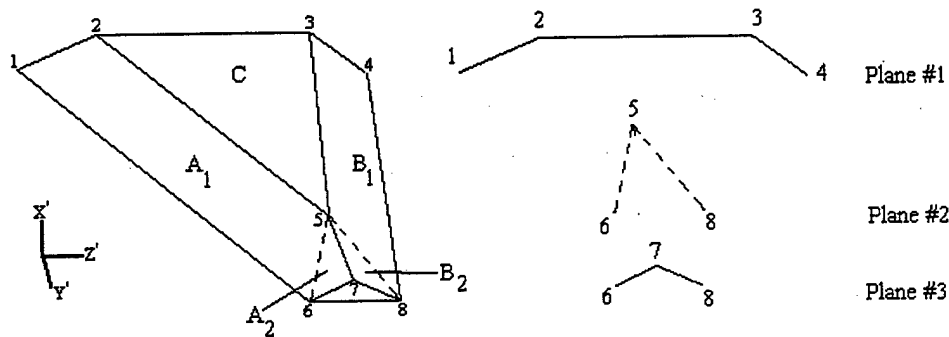


FIGURE 15. SAMPLE 2—USER DEFINED CONTROL SURFACE PARAMETERS

From the above figure it is seen that the surfaces ' $A_1$ ' and ' $A_2$ ' are not equivalent planar surfaces (i.e., do not have the same unit outer normal). A similar observation is made of the trailing edge of the fin (surfaces ' $B_1$ ' and ' $B_2$ '). Therefore, the above surface cannot be described using only Plane #1 and Plane #3 shown above. Plane #2 must be included to properly describe the control surface.

Note: Each plane must contain the same number of nodal points. Plane #1 contains four nodal points; therefore, Planes #2 and #3 must also contain four points. To meet this constraint, simply enter the center point for planes '2' and '3' twice (i.e., duplicate points '5' and '7').

Entry of the 'User Defined Geometry' is performed in the following dialog box. First, enter all of the 'x' points for a given plane, then all of the 'y', and finally, all of the 'z' points. Click the 'Next Plane' button to continue the process until all planes have been entered. Click on 'Finished' after entering all of the geometry points.

Control Surface Definition

(Plane #1 of 2)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e.  $z=0$  is the axial location of the hinge line)

X points: 0.0 .002225 0.0

Y points: .01524 .01524 .01524

Z points: -.0422 0.0 .032

(Use spaces between points)

Next Plane Delete Plane Finished

FIGURE 16. USER DEFINED CONTROL SURFACE INPUTS

- 'x', 'y', 'z' points - Local fin coordinates used for specifying the control surface geometry. 'x' is the fin thickness, 'y' is the radial distance measured from the missile surface, and 'z' is the axial distance measured relative to the hinge line.

- *Planes* - Displays the current plane as well as the maximum plane number.

### 3.3.4 Modified Wedge

If the control surface is a modified wedge, this section can be used in place of the 'User Defined Geometry' to simplify the geometric inputs required by the user.

Modified Wedge Control Surface	
Enter the leading edge half angle	3.018 [deg]
Enter the trailing edge half angle	3.977 [deg]
Enter the leading edge sweep angle	45 [deg]
Enter the span (root to tip distance)	0.0597
Distance from leading edge to hinge line	0.0422
	Root      Tip
Enter the leading wedge thickness	0.002225      3.822e-4
Enter the trailing wedge thickness	0.002225      5.04e-4
Enter the chord	0.0742      0.0145
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Convert to 'User Specified Geometry'"/>	

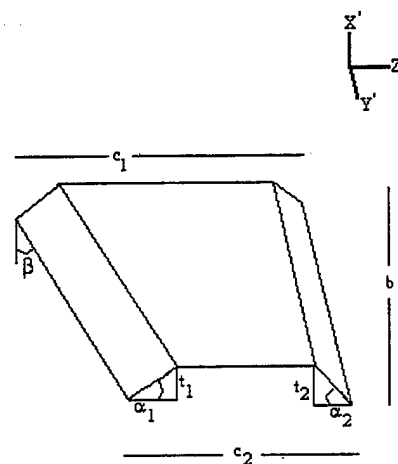


FIGURE 17. MODIFIED WEDGE CONTROL SURFACE PARAMETERS

- *Half Angles* - Enter both the leading and trailing edge half angles ( $\alpha_1$  and  $\alpha_2$ , respectively).
- *Sweep Angle* - Enter the leading edge sweep angle ( $\beta$ )
- *Span* - Enter the control surface span ( $b$ )
- *Distance to hinge line* - Enter the distance (on the root chord) from the leading edge to the hinge line.
- *Wedge thickness* - Enter the leading/trailing edge wedge thickness at both the root and the tip of the control surface ( $t_1$  and  $t_2$ , respectively).
- *Chord* - Enter the root and tip chords ( $c_1$  and  $c_2$ , respectively)
- *Convert* - When entering a control surface, it is often desirable to make modifications to the types of control surfaces allowed in Zeus<sup>++</sup>. If the control surface under consideration is similar to a modified wedge then start by entering all of the aforementioned parameters, then press the 'Convert to User Specified Geometry' button to convert the wedge parameters to 'x', 'y', 'z' coordinates.

It is then possible to modify the individual points to obtain the exact control surface desired.

### 3.3.5 NACA00xx

This section is used to describe a symmetric NACA00xx airfoil. Note that due to the bluntness of the leading edge, difficulty will most likely be encountered if this fin is evaluated in the Euler solver. It can, however, be used to generate a Navier-Stokes grid.

NACA 4-Digit Airfoil Control Surface

Enter the number of points used to describe the airfoil: 100

Enter the maximum thickness (t/c): 12 [% chord]

Enter the leading edge sweep angle: 45 [deg]

Enter the span (root to tip distance): 3.5

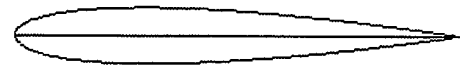
Enter the root chord: 2.0

Enter the tip chord: 1.5

Normalized distance from leading edge to hinge line (h/c): 1.0

OK Cancel

Convert to User Specified Geometry



NACA 0012 Symmetric Airfoil

FIGURE 18. NACA 4-DIGIT AIRFOIL PARAMETERS

- *Number of points* - Enter the number of points used to describe the airfoil geometry. Note that increasing this number will not significantly increase the computational time. It is simply the number of points used to draw the two-dimensional airfoil curve.
- *Thickness ratio* - Enter the thickness ratio as a percentage of the chord. (i.e., 12% refers to a thickness of  $0.12 \times \text{chord}$ )
- *Sweep Angle* - Enter the leading edge sweep angle in degrees.
- *Span* - Enter the span (i.e., distance from root to tip).
- *Root/tip chord* - Enter both the root and tip chords.
- *Distance to hinge line* - Enter the normalized distance ( $h/c$ ) from the leading edge to the hinge line (0-1) along the root chord.
- *Convert* - When entering a control surface, it is often desirable to make modifications to the types of control surfaces allowed in Zeus<sup>++</sup>. If the control surface under consideration is similar to a NACA 4-digit airfoil then start by

entering all of the aforementioned parameters, then press the 'Convert to User Specified Geometry' button to convert the fin parameters to a set of 'x', 'y', 'z' coordinates. It is then possible to modify the node points to obtain the exact control surface desired.

The following is the equation used to calculate the symmetric NACA00xx airfoils (where xx refers to the maximum thickness ratio):

$$\bar{r} = t \left( a_0 \bar{z}^{0.5} - a_1 \bar{z} - a_2 \bar{z}^2 + a_3 \bar{z}^3 - a_4 \bar{z}^4 \right)$$

$$a_0 = 1.4845, \quad a_1 = 0.6300$$

$$a_2 = 1.7580, \quad a_3 = 1.4215$$

$$a_4 = 0.5075$$

$$\bar{r} = \frac{r}{c}, \quad \bar{z} = \frac{z}{c}$$

'r' is the local radius, 'z' is the axial distance, 'c' is the chord, and 't' is the airfoil thickness ratio (thickness/chord).

### 3.3.6 Linked Fins

It is often desirable to consider different types of control surfaces at a given axial location (i.e., different size vertical and horizontal stabilizers). Zeus<sup>++</sup> implements this feature by linking two sets of control surfaces together, as described by the following example.

1. Define the first control surface. For example purposes create a 4 finned configuration oriented in the '+' configuration (i.e., fins at 90°, 180°, 270°, and 360°).
2. Delete the 90° and 270° fins on the first control surface.
3. Define a second control surface.
4. Delete the opposing fins on the second control surface (i.e., the 180° and 360°).
5. Select the "Overlap Previous Control Surface" dialog button for the second set of control surfaces.

### 3.4 GRID GENERATION

#### Menu Options

In this section, two separate computational domains are examined: the Euler grid and the Navier-Stokes grid. The Euler grid is the one used by the Zeus<sup>++</sup> tool (see notes below) for the inviscid and boundary layer solvers. The Navier-Stokes grid is generated, along with the appropriate boundary condition files; however, this grid is not used by the Zeus<sup>++</sup> tool in the flowfield analysis. Rather, this grid (and bc file) is created for use in a full Navier-Stokes solver. A number of menu options are available and are explained below. The first deals with the inviscid Euler grid and the remaining three pertain to the Navier-Stokes grid.

<u>Euler Grid</u>	Generate and view the Euler grid
<u>Generate N-S Grid</u>	Generate the Navier-Stokes grid and boundary condition data files 'filename\RunGrid\GASPGrid.p3da' & 'GASPBC.inp'
<u>Normal Spacing</u>	View the radial spacing (at the first point off the wall) along the missile
<u>View N-S Grid</u>	Plot the Navier-Stokes grid.

#### 3.4.1 Euler Grid

3.4.1.1 View Grid (Optional). Before running the inviscid code, it is often desirable to examine the computational domain to ensure that the relevant geometric features are spatially resolved and to ensure that the control surfaces are input correctly. Two methods of viewing the computational domain are available and are shown in the dialog box below. In the first, the grid is viewed as a series of two-dimensional planes (see sample grid below). The second method displays the Euler grid in three-dimensional block format.

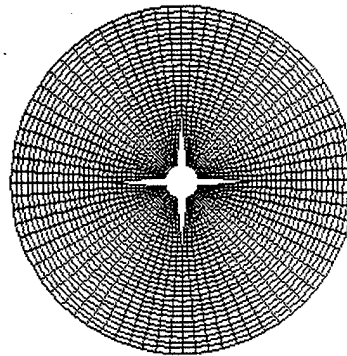
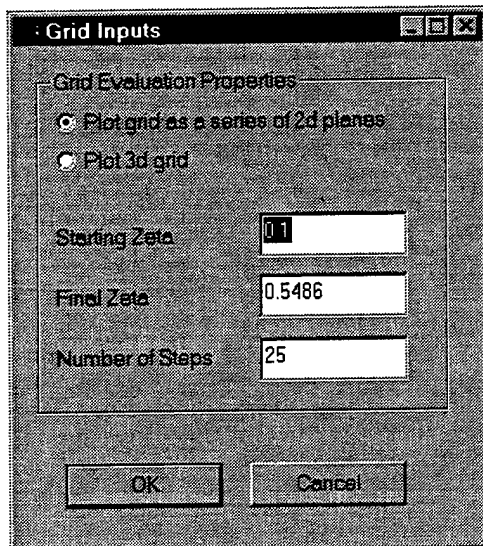
When using the '2d planes' option, it is possible to generate a movie of the grid that steps axially along the missile. To generate the movie file from the two-dimensional grid, run the Tecplot macro 'Playback\2dGrid\_Movie.mcr'. The movie file is written to the file 'Playback\Zeus\_Grid2d.rm' and can be replayed using the Tecplot Framer program.

Note: 1) The axial stepsize taken during the inviscid computation is controlled by the Courant-Friederichs-Lewy (CFL) stability condition specified in the integration parameter section. This is calculated at run-time, and therefore, the axial steps taken during the grid generation are not equivalent to those taken during the actual solution. For the purpose of evaluating the computational domain, equal axial spacing is taken along the missile.

2) There are two possible methods for generating the outer boundary of the computational domain. In the first, it is specified in terms of body radii in the grid parameter section. In the second, the outer boundary tracks the conical shock as the solution proceeds downstream. For the latter method, the conical shock location, and thus the outer boundary is not

known until runtime, and therefore, for the purposes of grid generation, the first method of calculating the outer boundary will be employed.

3) Viewing the Euler grid is not required before executing the inviscid solution. The actual grid used during the inviscid computation is calculated at runtime and can be viewed using the post-processing features of the Zeus<sup>++</sup> code.



2d Plane

FIGURE 19. EULER GRID GENERATION PARAMETERS

- *Plot type* - Generate the grid as either a series of two-dimensional planes (i.e., each plane is written as a separate Tecplot<sup>9</sup> zone), or a one three-dimensional grid (i.e., block format).
- *Starting/final zeta* - Enter the starting and final axial locations for the grid generation. Any subsection can be viewed using the grid generator, and therefore, the zeta values do not have to correspond to the starting/final axial locations of the missile.
- *Number of steps* - Enter the number of axial steps to be taken. The axial stepsize is assumed constant for the purposes of grid generation and is calculated linearly using the starting and final zeta values specified above.

### 3.4.2 Navier-Stokes Grid

**3.4.2.1 Generate Grid.** The Zeus<sup>++</sup> tool can be used to generate solutions for missile type geometries using both an Euler and a boundary layer solver. However, it is often desirable to perform a more in-depth analysis using a full Navier-Stokes solver. For these cases, the Zeus<sup>++</sup> tool serves as a grid generator, allowing the user to create complex three-dimensional grids in a matter of minutes. Currently, the tool is setup to generate CFD grids in Plot3d format ('filename\RunGrid\GASPGrid.p3da') along with the appropriate zonal boundary conditions ('filename\RunGrid\GASPBC.inp') for the GASP<sup>1</sup> flow solver. The boundary condition file specifies each point on a zone edge as either a solid surface (a fin) or a zonal boundary.

# of Z stations	Ending Z location	Beginning Spacing	Ending Spacing
20	.0686	.003	.005
12	.230	.01	.003
20	.3064	.003	.003
12	.4775	.003	.003
21	.5486	.003	.003
20	.7	.001	-NA-

FIGURE 20. NAVIER-STOKES GRID GENERATION PARAMETERS

- *Axial clustering* - In this section the axial clustering for the Navier-Stokes grid is specified. The radial and circumferential values are declared in the grid parameter section.

Note: 1) Zeus<sup>++</sup> uses a Newton method to calculate the axial clustering parameters (i.e., geometric stretching factors). To prevent a divergent solution, a valid initial spacing must be provided. This is accomplished by first specifying the number of axial sections along the missile. Next, enter the number of points and the ending axial location for each section. Finally, click on the 'Reset Spacing' button to calculate the appropriate parameters for an equally spaced grid. Once a valid grid has been obtained, the beginning/ending spacing can be adjusted incrementally to obtain the desired axial clustering.

2) The grid and boundary condition output files produced during the grid generation process are often times very large, and therefore, may take some time to generate. Also, the process of determining the appropriate axial clustering is an iterative procedure, and is independent of the circumferential layout of the grid. Therefore, to minimize the amount of calculation required per iteration, set the number of circumferential points to unity in the Navier-Stokes grid section. After the desired axial clustering is obtained, reset the number of circumferential points to the desired value, and regenerate the final Navier-Stokes grid.

- *Number of axial stations* - Specify the number of axial sections in which clustering is desired. For the sample grid shown below the six sections designated are distinguished by the vertical tic marks. The division shown in the example allows for axial clustering at the beginning/end of the nose section, leading/trailing edges of both sets of fins, as well as clustering at the aft end of the missile.
- *Number of 'z' stations* - The number of axial points per section.
- *Ending 'z' location* - Ending axial location for each section [ft/m].
- *Reset spacing* - Reset the beginning/ending axial spacing in each section to obtain equal spacing in all sections.
- *Beginning spacing* - Beginning axial spacing ( $\Delta z$ ) for each section [ft/m].
- *Ending spacing* - Ending axial spacing ( $\Delta z$ ) for each section [ft/m].
- *Aft grid* - If a grid aft of the missile is desired (subsonic flow) then this checkbox must be selected.
- *Number of radial points* - This refers to the points which lie radially between the missile centerline and the missile surface directly behind the missile (see yellow grid section or area designated by the horizontal tic marks). The radial spacing at the first point is taken to be the same as that on the surface (at the aft end) of the missile. Zeus<sup>++</sup> calculates the geometric stretching parameter that places the final radial point along the missile centerline. If the error message 'Unattainable radial clustering' is displayed, try reducing the number of radial points in the aft grid section.
- *Outer boundary clustering* - Specify the outer boundary clustering around the nose section of the missile.
- *First point angle* - The angle between the missile centerline and the first nose point [degrees]. See angle 'a' below. *The angle must be greater than 0 degrees.*
- *Last point angle* - The angle between the missile centerline and the final nose point [degrees]. See angle 'b' below. *The angle must be less than 90 degrees.*

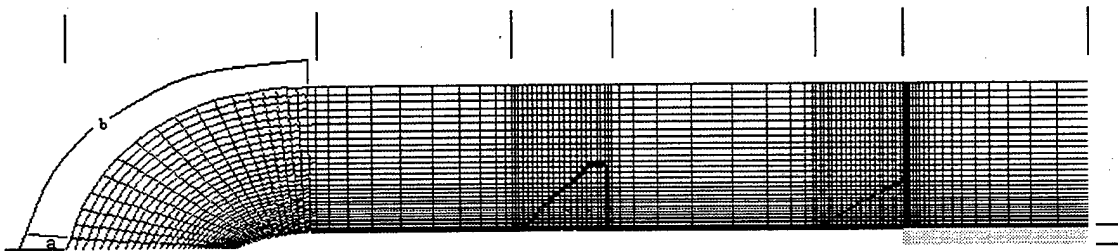


FIGURE 21. SAMPLE CROSS-SECTION OF NAVIER-STOKES GRID

The final step in generating a Navier-Stokes grid is to specify the surface boundary conditions (see figure below).

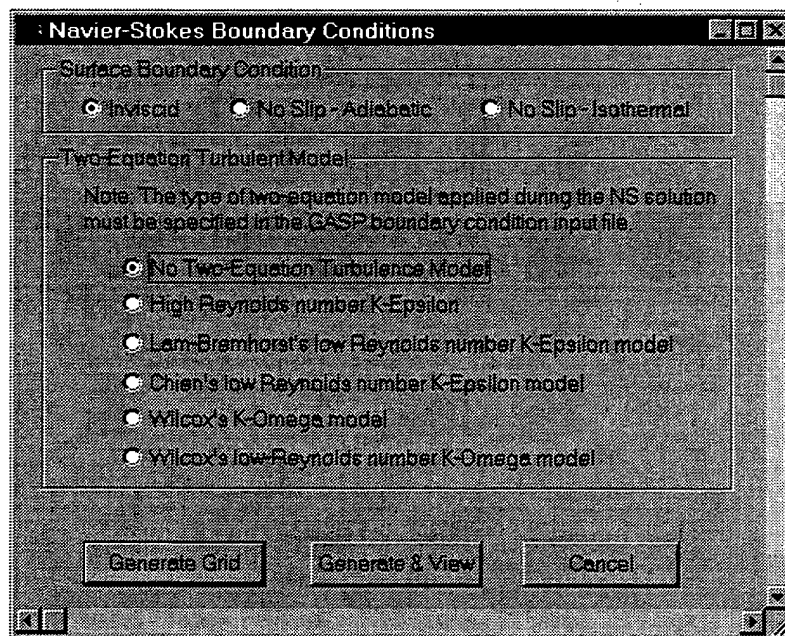


FIGURE 22. NAVIER-STOKES BOUNDARY CONDITION PARAMETERS

- *Surface boundary condition* - Specify the type of boundary condition to be employed when a solid surface is encountered. Currently, the Zeus<sup>++</sup> tool generates a boundary condition file for the GASP flow solver, which specifies the boundary condition along the zone edges (i.e., solid surface or zonal boundary).
- *Two-Equation Model* - The type of two-equation turbulence model applied during the Navier-Stokes solution must be specified in the GASP boundary condition input file.

**3.4.2.2 View Normal Spacing.** A prerequisite for accurately computing a Navier-Stokes solution is the spatial resolution of the boundary layer, and therefore, the computational grid must be appropriately refined near the surface. The boundary layer thickness cannot be determined a priori, however, an approximate value can be obtained using flat plate assumptions. Using this estimate for the boundary layer thickness, the radial clustering parameter can be increased/decreased until an appropriate value for the normal spacing to the first point off the surface is obtained. A typical value for the required normal spacing is 1/20th to 1/30th of the approximate boundary layer thickness (i.e., approximately 20-30 points in the boundary layer).

**3.4.3.3 View Grid.** This option plots the Navier-Stokes grid in three-dimensional block format using the Tecplot<sup>9</sup> plotting package. A two-dimensional slice of the three-dimensional grid can be obtained by executing the Tecplot macro 'Playback\GaspGrid.mcr'. This is the view displayed in the Generate Gasp Grid section,

and is useful for evaluating the axial clustering parameters as well as the spatial resolution of the control surfaces.

Note: The control surfaces are represented by the contour plot variable 'onfin'. Unity and zero represent a solid surface and a zonal boundary, respectively. To evaluate the spatial resolution of the control surfaces, examine a contour plot of the 'onfin' parameter.

## CHAPTER 4: INPUT PARAMETERS

### OVERVIEW

A variety of parameters are required before execution of the Euler or boundary layer solvers can proceed. The runtime parameters are specified in the 'Inputs' section from the main menu. The parameters are separated into four basic categories as shown below.

1. Aerodynamic Data
  - Run Matrix
  - General Data
2. Integration Controls
3. Separation Parameters
4. Boundary Layer Controls
  - Execution Parameters
  - Modeling Parameters
  - Output Parameters
5. Output Controls

### 4.1 AERODYNAMIC PARAMETERS

#### 4.1.1 Run Matrix Parameters

The Aerodynamic run matrix is where the Mach numbers and missile orientations are specified. This is accomplished by entering an initial, final, and incremental value for Mach number, angle of attack, angle of yaw, and roll angle (see figure below). Zeus<sup>++</sup> then calculates an Aerodynamic run matrix consisting of all possible cases.

The screenshot shows a software dialog box titled "Aerodynamic Run Matrix" with a "General Conditions" tab. It contains several input fields for defining sweep parameters:

- Mach Number Sweep:**
  - First Mach Number: 4.0
  - Last Mach Number: 4.5
  - Mach Number Intervals: 0.25
- Configuration Roll Position Sweep:**
  - First Roll Position: 0
  - Last Roll Position: 45
  - Roll Position Intervals: 22.5
- Angle of Attack Sweep:**
  - First Angle of Attack: 0
  - Last Angle of Attack: 20
  - Angle of Attack Intervals: 10
- Deflection Sweep:**
  - First Tail Deflection: 0
  - Last Tail Deflection: 1
  - Tail Deflection Intervals: 1
- Angle of Yaw Sweep:**
  - First Angle of Yaw: 0
  - Last Angle of Yaw: 4
  - Angle of Yaw Intervals: 4

At the bottom right of the input area is a button labeled "Default Values". At the very bottom of the dialog are four buttons: "OK", "Cancel", "Apply", and "Help".

FIGURE 23. AERODYNAMIC SWEEP PARAMETERS

- *Mach number sweep* - Enter the starting, final, and incremental Mach number. For the sample shown above, a Mach number of 4.0, 4.25, and 4.50 will be considered.
- *Angle of attack sweep* - Enter the starting, final, and incremental angle of attack.
- *Angle of yaw sweep* - Enter the starting, final, and incremental yaw angle.
- *Roll angle sweep* - Enter the starting, final, and incremental roll angle.
- *Deflection sweep* - Running sweeps on control surface deflections has not been implemented in the current version.

**Note:** For each case (i.e., every Mach number, angle of attack, angle of yaw, and roll angle combination) a subdirectory will be created under the main directory and will be labeled Run0001-Runxxxx. In addition, a file called RunMatrix.txt is generated which maps the case number to the aerodynamic parameters entered above.

#### 4.1.2 General Aerodynamic Parameters

A number of general aerodynamic parameters must be specified before executing either the Euler or boundary layer solvers. These parameters are shown in the following figure and described in detail below.

The screenshot shows the 'Aerodynamic Run Modes' dialog box with the 'General Conditions' tab selected. The 'Freestream Data' section has two radio buttons: '1962 U.S. Standard Tables' (selected) and '1959 ARDC Tables'. Below them are input fields for 'Altitude' (0), 'Pressure' (8117.27), 'Density' (0.103877), and 'Gamma' (1.4). The 'Units' section has two radio buttons: 'English' (selected) and 'Metric'. The 'Reference Conditions' section has input fields for 'Reference Diameter' (0.03048), 'Reference Area' (0.00073), and 'Axial Moment Reference' (0.2887). The 'Viscous Parameters' section has two radio buttons: 'Adiabatic' (selected) and 'Wall temperature' (500), and a 'Prandtl number' (0.72). The 'Gas Type' section has two radio buttons: 'Perfect Gas' (selected) and 'Real Gas'. At the bottom are 'OK', 'Cancel', 'Apply', and 'Help' buttons.

FIGURE 24. GENERAL AERODYNAMIC INPUT PARAMETERS

- *Free-stream data* - The required free-stream data includes the pressure, density, and ratio of specific heats ( $\gamma$ ). There are three methods of specifying these parameters.
  - 1) Select either of the standard atmospheric tables (1962 U.S. Standard Tables or 1959 AFDC Tables) and then enter an altitude in the provided dialog box. The pressure and density will be calculated from the standard table data and the ratio of specific heats is assumed that of standard air (1.4).
  - 2) Specify the pressure, density, and ratio of specific heats directly by clicking on the radio button next to these parameters and entering the desired values.

- 3) Specify a unit Reynolds number and let Zeus<sup>++</sup> calculate the corresponding pressure and density. Gamma, the ratio of specific heats, is taken to be that of standard air (1.4).

$$Re_L = \frac{\rho_\infty U_\infty}{\mu_\infty}$$

- *Units* - Specify either English or metric units.

Note: All of the parameters entered must be in base units (i.e., meters, not centimeters; feet, not inches, ...etc.). Incorrect results will be obtained if you enter all of your lengths in centimeters and then try to compensate by entering the reference quantities in centimeters. Although entering non-standard units will not effect the inviscid solution, it will produce incorrect results for the viscous solution due to the Reynolds number effect.

- *Reference conditions* - Enter the reference diameter (typically the base diameter) and the reference area (typically the area calculated using the base of the missile). Also, specify the axial location about which the pitching and yawing moments are calculated (The rolling moment is always taken about the missile centerline).

Note: A nose down pitching moment is negative.

- *Viscous parameters* - If the viscous boundary layer code is to be considered then the wall boundary condition for the heat flux and the Prandtl number must be specified. For the adiabatic wall, calculating an appropriate wall temperature using the recovery factor methodology imposes a zero heat flux boundary condition.
- *Gas type* - Select either a perfect gas or a real gas solution.

Note: If 'real gas' is selected then the approximate Riemann solver must be selected in the integration controls section.

## 4.2 INTEGRATION CONTROLS

### 4.2.1 Parameter Description

A number of user specified integration parameters must be specified in the following section. These include axial limits for integration, starting solution, stability criterion, limiters, and flux calculation parameters.

FIGURE 25. NUMERICAL INTEGRATION CONTROLS

- *Space marching* - Enter the starting and final axial locations for the solution. Also, enter a maximum number of steps to take before the code aborts.

Note: The starting axial location is dependent on the type of starting solution applied (see below for an explanation of each of the different methods of generating a starting solution).

- *Stability criterion* - Enter a step size safety factor (analogous to a CFL number) between zero and unity. Also, specify a maximum allowable step size to ensure that all relevant features are spatially resolved in the axial direction.
- *Starting solution* - The Euler solver is a marching code, and therefore, a starting solution must be specified on the missile before the integration can proceed. Three different types of starting solutions are available, each of which is discussed below.
  - 1) *Free-stream* - A free-stream profile is specified as the starting solution at the axial location entered in the 'space Marching' section. For this type of inflow profile, the outer boundary does not track the conical shock, but rather, is user specified in terms of body radii in the grid parameter section.

Note: The region upstream of this profile is ignored, and therefore, in order to accurately compute the forces and moments, the integration should begin as close to the nose as is numerically possible.

2) Cone Solution - The cone starting solution applies a one-dimensional cone solver to the region from the nose to the beginning axial location specified in the space marching section. This calculates the shock angle, as well as the flowfield directly aft of the shock, given the local turning angle. The starting profile is then taken to be that generated by the conical solver. In other words, a conical flow solver is applied from the nose tip to the location specified as the 'Axial Location to begin computation'. The Euler solver then begins integration at this location with the profile supplied by the conical flow solver.

Note: a) The conical flow solver is intended to provide the shock angle as well as the flowfield properties directly aft of the shock. It is not intended to solve the flowfield for the entire nose cone. The one-dimensional cone solver cannot calculate the forces/moments, and therefore, if you do not start the Euler solver until the end of the nose section, the forces/moments due to this section will be neglected. In other words, the conical solver should only be applied to a very small region around the nose tip and not to the entire nose section.

b) If the local turning angle becomes excessively large then the one-dimensional cone solver will compute a subsonic Mach number aft of the shockwave. The Zeus Euler solver is a space marching code, and therefore, cannot handle subsonic regions. For this case, a free-stream starting solution must be applied.

3) Blunt body - For a blunt body, the flow will be subsonic in some region surrounding the nose section. If you attempt to begin the Zeus integration too close to the nose, the code will encounter a subsonic region and abort. To prevent this, a blunt body solver must be applied to the entire subsonic region. This solver provides a solution from the beginning of the nose to the value specified in the space marching section labeled 'Axial location to begin computation'. The user must ensure that this value is downstream of the entire subsonic region.

Note: In contrast to the free-stream and conical starting profiles, the blunt body solver calculates the forces and moments in the region between the nose and the starting location for the Euler solver. These forces and moments are summed with the values obtained by the Euler solver to provide the total forces/moments acting on the body. Thus, for this starting procedure, it is acceptable to run the blunt body solver over the entire nose section.

- *Riemann solver* - Specify whether the full Riemann or approximate Riemann solver is applied.
- *Bow shock flux* - Calculate the numerical flux at the bow shock using either the complete Riemann solver or freestream properties.

- *Bow shock angle* - Calculate the bow shock angle using either the complete Riemann solver or the Roe averaged variables.
- *Interior point limiter* - Numerical limiter at all interior points (i.e., not adjacent to a surface). The default value is unity, however, this can be increased/decreased to modify the numerical damping provided by the integration scheme. The applicable range is 2 to 0, where 2 and 0 are the least and most dissipative, respectively. A value of zero reduces the integration scheme to a first-order Gudonov method.

Note: Decreasing the damping (i.e., increasing the limiter value above) will improve the accuracy of the numerical scheme and will more accurately capture the flowfield discontinuities. Insufficient damping can often lead to numerical oscillations in the flowfield and prevent a stable integration.

- *Turning angle correction* - If a subsonic region is encountered while turning the flow, the Zeus code will apply a limiter known as a turning angle correction. The actual turning angle encountered in the flow is multiplied by the turning angle correction to try to prevent a subsonic region. If this correction does not prevent a subsonic region then the code will abort. Typically, a value of 0.9 (i.e., 90% of the original angle) is applied.

## 4.3 SEPARATION PARAMETERS

### 4.3.1 Parameter Description

The Zeus code is an inviscid solver, and as such, neglects the effects of viscosity on the flowfield. Since flow separation is a viscous phenomenon, an Euler solver alone cannot accurately predict the effects of a separation zone on the flowfield. Two separation models (clipping and forced) are available in the Zeus Euler solver which attempt to improve the predictive capabilities of the tool near separation zones.

The clipping separation model is described in detail in Ref.11 and operates by decreasing the crossflow velocity on and near the body surface. The crossflow velocity reduction is accomplished by setting an upper limit to the allowable crossflow velocity. If the velocity at any point exceeds this value, it is reduced to this level. Pressure and density are assumed unchanged and the axial component of velocity is increased to give the correct stagnation enthalpy value. Clipping destroys the crossflow shock and produces a large vortex on the leeside of the body that is in qualitative agreement with experiment. Computed pressures on the leeside of the body are in better agreement with experiment. However, clipping tends to increase the windward pressures and often decreases the accuracy of the results in this region.

The forced separation model, originally described in this form by Ref. 12, seeks to simulate separation by altering the velocity direction along a user defined separation line. In each crossflow plane, the separation model is applied to wall cells "s" and "s+1", which

are immediately windward and leeward of the separation line, respectively. Pressure and density at these points are defined by:

$$p_s = 0.5(p'_{s-1} + p'_{s+1})$$

$$\rho_s = 0.5(\rho'_{s-1} + \rho'_{s+1})$$

$$p_{s+1} = 0.5(p'_s + p'_{s+2})$$

$$\rho_{s+1} = 0.5(\rho'_s + \rho'_{s+2})$$

Here ' denotes old values. The flow velocities are determined by prescribing a streamline direction and assuming constant stagnation enthalpy. Unfortunately, solutions obtained with the forced separation model are sensitive to mesh size. As the mesh is refined, the pressure beneath the vortex diminishes.

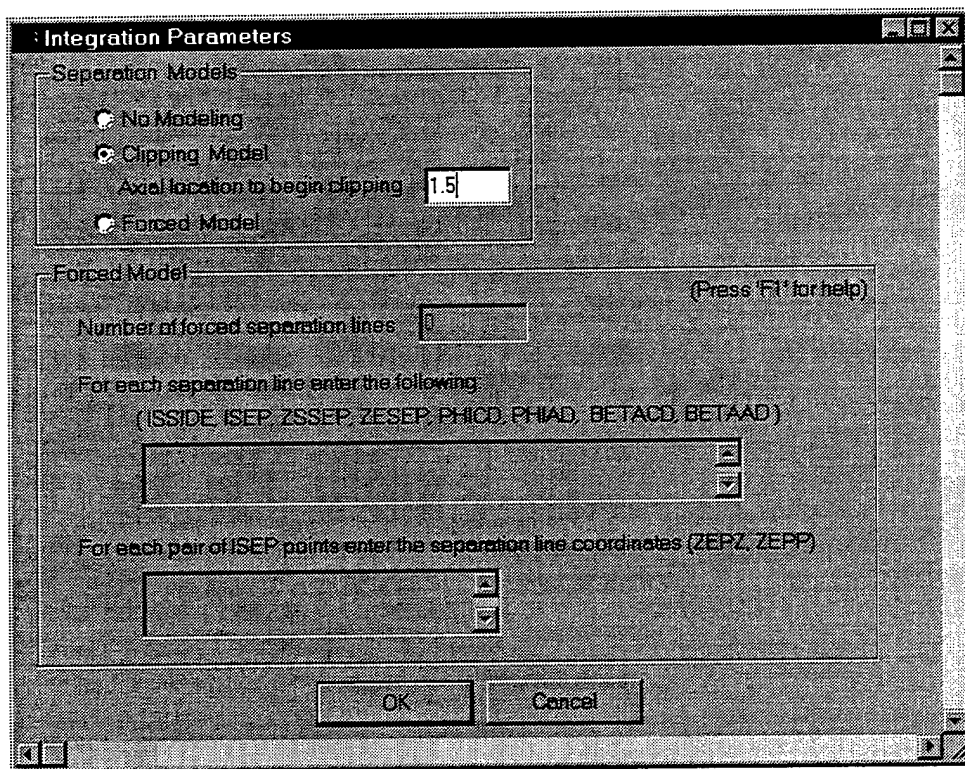


FIGURE 26. SEPARATION MODELING PARAMETERS

- *Separation Model* - Select the desired separation model.
- *Clipping Separation* - Enter the axial location to begin clipping.
- *Forced Separation* - Enter the number of forced separation lines. For each of the lines, the user must specify the following parameters:
  - *ISIDE* - The side of the body on which the separation occurs. Zero if separation occurs between 0° and 180°, unity if separation occurs between 180° and 360°.

- *ISEP* - The number of points used to define the separation line.
- *ZSSEP* - Zeta value at which separation is started.
- *ZESEP* - Zeta value at which separation is terminated.
- *PHICD, PHIAD, BETACD, BETAD* - Flow direction in degrees. Typical values are 20°, 20°, 20°, 5°, respectively.
- *ZEPZ, ZEPP* - A list of ISEP pairs of separation line coordinates where ZEPZ and ZEPP are the z and  $\phi$  coordinates for each point.

#### 4.4 BOUNDARY LAYER CONTROLS

##### 4.4.1 BL Execution Parameters

The boundary layer code requires, as a boundary condition, the inviscid solution from the Zeus code. Therefore, before executing the boundary layer code, Zeus<sup>++</sup> must save the inviscid solution at a number of different axial stations. It is both unnecessary and infeasible (due to computational constraints) to save the inviscid flowfield at every axial station, however, an adequate number of axial stations must be considered in order to accurately solve the boundary layer equations. The user must save the inviscid solution often enough to resolve all relevant geometric features axially (i.e., must have several axial stations written out on each control surface).

- *Output inviscid flowfield* – Enter the increment and the axial stepsize for saving the inviscid flowfield. The Zeus code will save the inviscid flowfield every incremental axial step as well as every time the axial stepsize exceeds the specified value.

Note: The Zeus<sup>++</sup> code must save the inviscid flowfield often enough to axially resolve all relevant geometric features.

- *Starting axial location for integration* – Enter the axial location to begin the boundary layer solver. Note that this must lie downstream of the starting location of the Euler solver as specified in the Integration Parameters section.
- *Minimum number of steps* – Enter the minimum number of axial steps the boundary layer solver will take. The code uses this parameter to calculate a maximum allowable stepsize.
- *Maximum number of steps* – Enter the maximum number of axial steps the boundary layer solver will take before aborting.

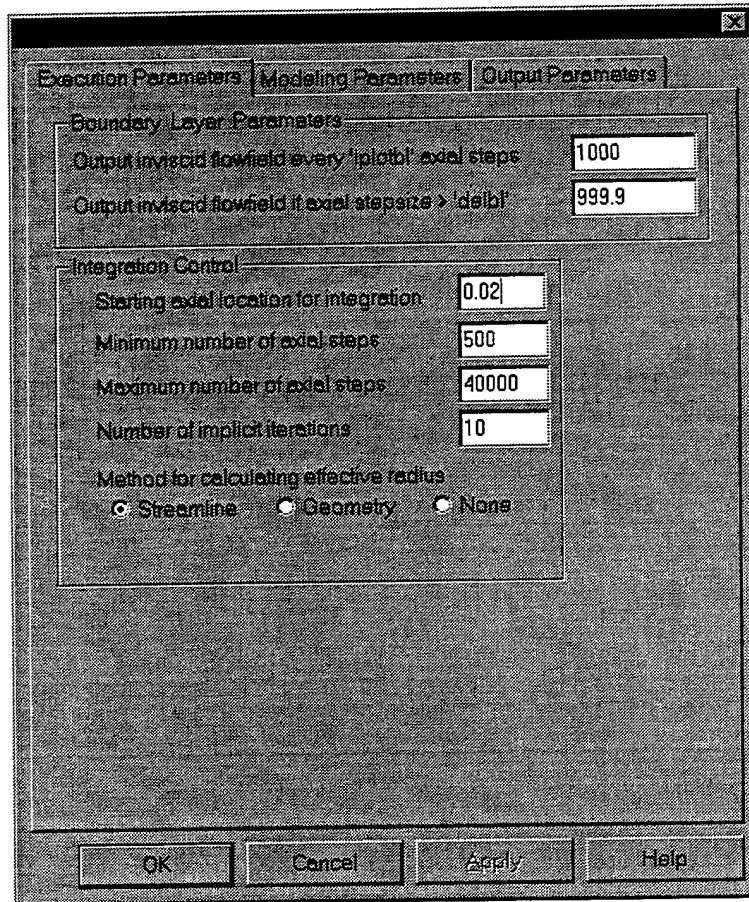


FIGURE 27. BOUNDARY LAYER EXECUTION PARAMETERS

- *Implicit iterations* – Number of implicit iterations.
- *Effective radius* – This parameter selects the manner in which the effective radius is calculated; the method by which streamline spreading is determined. The “streamline” option uses the distance between successive streamlines, the “geometry” option computes the local surface radius, while “none” sets the radius to unity. Optimal results are obtained with the “streamline” option except on circular bodies at zero incidence. Here the “geometry” option is recommended.

## 4.4.2 BL Modeling Parameters

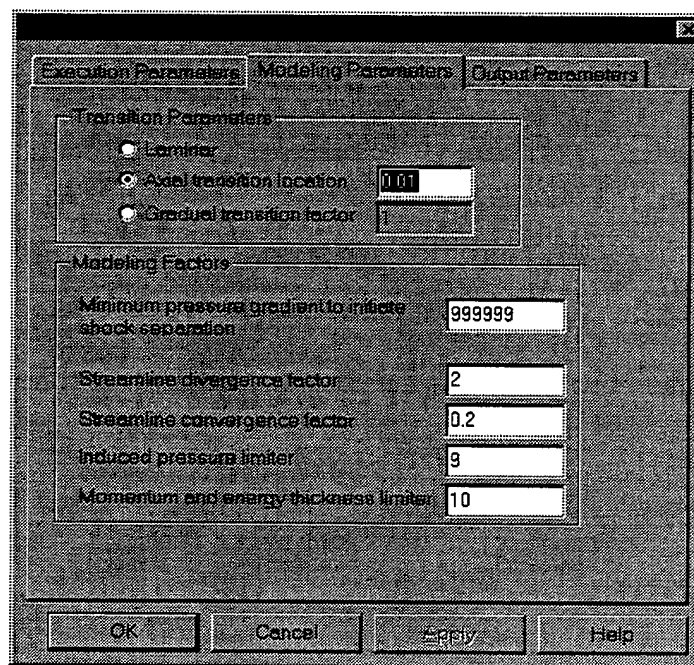


FIGURE 28. BOUNDARY LAYER MODELING PARAMETERS

- *Transition parameters* - Select one of the three possible transition parameters. If laminar is selected then the flow is forced to remain laminar for the entire calculation. If an 'Axial transition location' is specified, the flow is laminar up to this point and then abruptly transitions to turbulent flow. If the 'Gradual transition factor' is selected, then the transition model is taken from Ref. 6, and the parameter specified is  $F_t$  (default of 1.0) from Eq. 23.
- *Pressure gradient for separation* - If the streamwise pressure gradient exceeds this user specified parameter the boundary layer and energy thicknesses are reduced to simulate separation<sup>8</sup>.
- *Streamline divergence* - Controls streamline divergence as described in Reference 6 ( $K_d$  from Eq. 32). Set to unity to follow inviscid streamlines. Use a value of 2.0 for bodies at incidence. Only applies to edge #1 of each zone.
- *Streamline convergence* - Controls streamline convergence as described in Reference 6 ( $K_c$  from Eq. 32). Set to unity to follow inviscid streamlines. Use a value of 0.2 for bodies at incidence. Only applies to edge #1 of each zone.
- *Induced pressure limiter* - The boundary layer code attempts to correct for the induced pressure effects caused by the formation of the boundary layer. To prevent large pressure corrections around leading/trailing edges, the induced pressure is limited using the 'Induced pressure limiter' as discussed in Ref. 6 ( $K_r$  in section 4.3).

- *Momentum/Energy thickness limiter* - A parameter which limits the momentum/energy thickness in anticipated regions of separation. The limiter is discussed in Ref. 6 (k from Eq. 33).

#### 4.4.3 BL Output Controls

The following dialog box controls the output from the boundary layer solver.

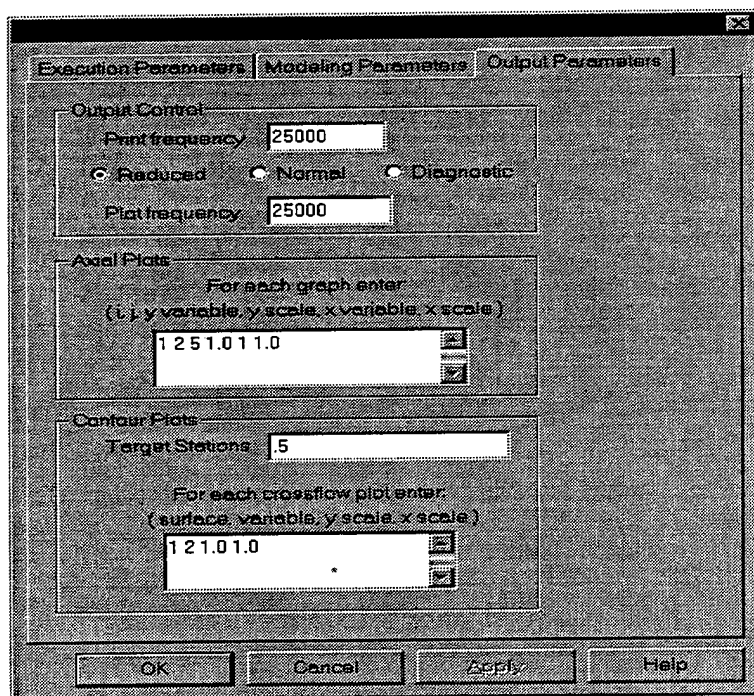


FIGURE 29. BOUNDARY LAYER OUTPUT PARAMETERS

- *Print Frequency* - The ASCII output table, 'ZeusBL.out' is generated every 'print frequency' steps. The 'reduced/normal/diagnostic' option controls the amount of data written to the output tape. "Reduced" prints only the most important quantities such as  $C_f$  and  $C_h$ . "Normal" includes additional boundary layer parameters as well as edge conditions.
- *Plot Frequency* - The contour plot tape is written every "plot frequency" steps.
- *Axial Plots* - Specify the surface number, point location, dependent and independent variables, as well as x/y scaling factors. Each of the axial plots will be written to a file 'graphxxx', where 'xxx' is the number of the axial plot.

i - surface number

j - point number

Y variable - Select one of the following for a dependent variable:

1. pressure
2. density
3. edge velocity
4. temperature
5.  $C_f$  (base on edge conditions)
6.  $C_f$  (base on reference conditions)
7.  $C_h$  (base on edge conditions)
8.  $C_h$  (base on reference conditions)
9. Momentum thickness
10. Displacement thickness
11. Energy thickness
12. Heating rate ( $B/(ft^2 \text{ sec})$ ,  $J/(m^2 \text{ sec})$ )
13. Viscous loads; Here "i" is the surface number and 'j' is the load (1=x force, 2=y force, 3=z force, 4=x moment, 5=y moment, 6=z moment). To get total body viscous loads set "i" to 0.
14. Induced loads. Same options apply as for viscous loads.

Y scale - Dependent variable scale factor.

X variable - Select one of the following for an independent variable:

1. Axial location, z
2. Reynolds number (based on axial location),  $Re_z$
3. Streamline length, s
4. Reynolds number (based on streamline length),  $Re_s$

X scale - Independent variable scale factor.

- *Contour Plots* - Specify a list of target stations (separated by blank spaces) where the contour plots are desired (the files are written to 'graphcxxx', where 'xxx' refers to the appropriate contour file number). Then list the following parameters for each contour plot:

Surface - Enter the surface number. The surface numbers are counted sequentially along each of the four edges for each zone (i.e., Zone 2, Edge3 corresponds to surface number 7).

Variable - Select one of the following for a dependent variable:

1. pressure
2. density
3. edge velocity
4. temperature

5.  $C_f$  (base on edge conditions)
6.  $C_f$  (base on reference conditions)
7.  $C_h$  (base on edge conditions)
8.  $C_h$  (base on reference conditions)
9. Momentum thickness
10. Displacement thickness
11. Energy thickness
12. Heating rate ( $B/(ft^2 \text{ sec})$ ,  $J/(m^2 \text{ sec})$ )

y-scale - Dependent variable scale factor.

x-scale - Crossflow variable scale factor.

## 4.5 OUTPUT CONTROLS

### 4.5.1 Parameter Description

This section controls which type of output the Zeus Euler solver generates, as well as the output frequency. The first section in the figure below controls the output for the surface pressures, forces, moments, and centers of pressure. The second section controls the output for generating contour plots of the flowfield variables. The final section controls both the ASCII output and screen residual.

**Output Controls**

**Surface Pressures - Forces - Moments**

☐ Print cell coordinates  
☒ Do not print cell coordinates

Write surface properties at every 'k' axial station: 50

Write surface properties if stepsize is greater than DELZA: 5000

**Contour Data**

Write contour properties at every 'k' axial station: 200

and

at the following user specified axial locations: 0.20 0.22 0.23 50 51 52  
 (Must be in descending order)

Write contour properties if axial stepsize is greater than DELZC: 5000

**ASCII Data**

Output residual every ISKIP iterations: 10

Print crossflow plane if step number is evenly divisible by IPRINT: 5000

Print N planes which are evenly divisible by NSKIP: 1

Print M planes which are evenly divisible by MSKIP: 1

OK Cancel

FIGURE 30. OUTPUT CONTROL PARAMETERS

- *Surface pressures, forces, and moments* – This section handles the output for the surface pressure, forces, and moments.
- *Print cell coordinates* – Write the cell coordinates (axial and radial location) at each cell where the data is written along with the surface pressure, forces, moments, and centers of pressure.
- *Write surface properties* – Specify both an incremental value and a maximum stepsize for generating the output data. The Zeus<sup>++</sup> code writes the output at every specified incremental value (axially) as well as if the axial stepsize is larger than the specified parameter.
- *Contour data* – Contour plots of a variety of flowfield parameters (pressure, density, Mach number, etc...) can be generated at various axial stations.
- *'k' Axial stations* – Enter the increment for generating the contour plot output data. The code will write the output data at every user specified incremental axial station.
- *User defined stations* – Specify the specific axial locations where the contour data is desired. This is useful for generating contour plots at specific locations in the flowfield (i.e., leading/trailing edges of control surfaces).
- *Given stepsize* – The Zeus<sup>++</sup> code generates a contour plot file every time the axial stepsize is greater than the specified value.
- *ASCII data* – The ASCII data is written to the 'fort.9' data file in the 'filename' directory.
- *Output residual* – Specify the axial step increment for writing out the residual data. Note, the code writes the residual data to the screen as well as to the 'fort.9' data file.
- *Crossflow plane* – Specify the axial increment for saving the ASCII data from a crossflow plane (i.e., an axial station).
- *Print 'M', 'N' planes* – Specify the radial and circumferential increments for saving the ASCII crossflow plane data (i.e., Write out the data at every 'm' radial points and 'n' circumferential points). The computational effort required for I/O can be minimized by not writing out every radial and circumferential data point.

## CHAPTER 5: CODE EXECUTION

### OVERVIEW

This section is used to select which cases are evaluated as well as what type of solution is performed. A step by step procedure for executing a run is provided below along with a sample view of the execution screen.

- Select the desired run.

Note: Multiple runs may be selected in a number of ways. The first is to hold down the 'Ctrl' key and click on each of the desired runs. Alternatively, you can click on the first desired run, hold down the 'shift' key, and then click on the last run to select all cases in-between. Finally, you can click on the 'select All' button to run all available cases.

- Select the type of run desired.

Note: If 'Full Run' is selected the code will execute the Inviscid Euler solver, write out the flowfield at the axial intervals specified in the Boundary Layer Execution Parameter section, and then execute the boundary layer solver to get an approximate viscous solution. If 'Inviscid Run' is selected the code will execute the Inviscid Euler solver and then terminate. No viscous approximation will be calculated. If a viscous calculation is desired, click on the 'Viscous Run' button, and select the desired case. Only the runs that have been solved with the inviscid solver will show up as choices when the 'Viscous Run' button is selected. This is because the viscous boundary layer solver requires the inviscid flowfield as a boundary condition.

- Specify whether a symmetry plane exists.

Note: If you know a priori that the geometry is non-symmetric about the pitch plane, you must select 'Full Grid (All Runs)'. If the geometry is symmetric about the pitch plane choose either 'symmetry Plane (All Runs)' if you will not be running any asymmetric cases, or alternatively, choose 'symmetry Plane (Zero Roll and Yaw)' to run a symmetry plane only for those cases with zero yaw and roll angles. If a symmetry plane is being considered then only half of the computational domain (180 vs. 360 degrees) is considered.

*Running with a symmetry plane will cut the computational time approximately in half.*

- Click on 'OK' to perform the desired calculations.

- After the runs have completed, revisit the 'Execution' section to check on the termination status of each run. If the 'Runs' column contains one asterisk (\*) then the inviscid solution has completed (e.g., Run0001). If it contains double asterisks (\*\*) then both the inviscid and viscous solutions have been completed (Run0002). In addition, the 'Exit Status' column should contain 'Normal Exit' if no runtime errors were encountered. If the code did not exit normally then an error code will be provided. These error codes are 5000 series numbers for the Euler solver, 6000 series numbers for the boundary layer solver, 7000 series numbers for the cone solver, and 8000 series numbers for the blunt body solver. A description of each of the possible error codes is presented in the Error Code section.

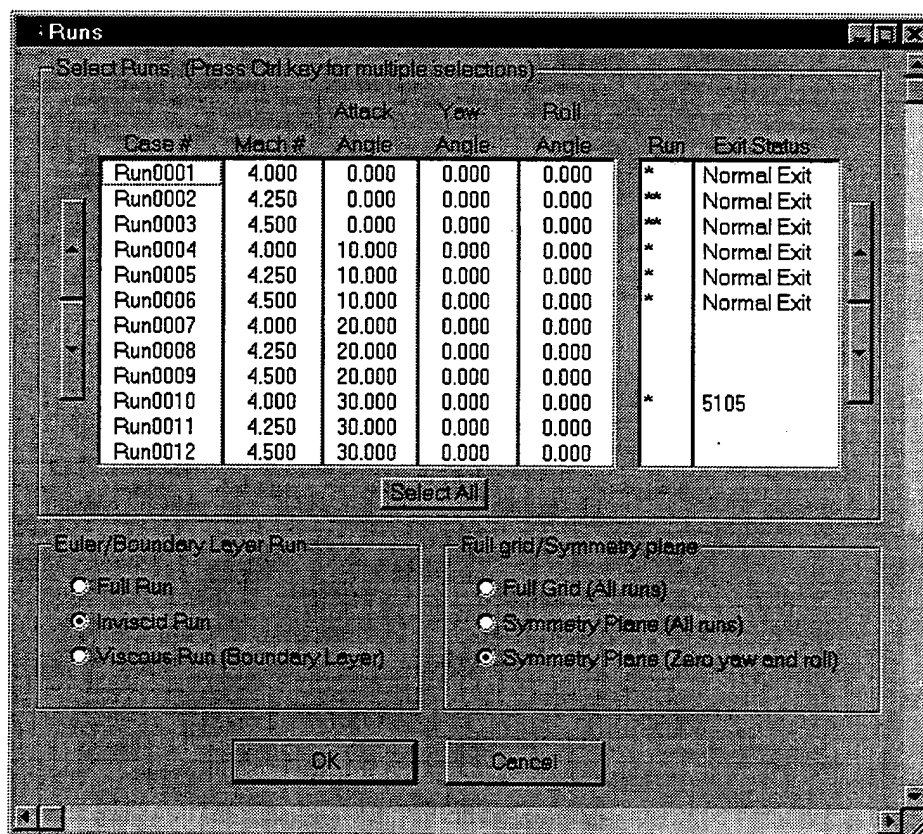


FIGURE 31. EXECUTIVE CONTROL PARAMETERS

## 5.1 CONSOLE OUTPUT

### 5.1.1 Euler Console Output

As the Euler solver is executing, a variety of information is written to a command prompt (see sample below). This includes messages denoting which code is currently running (cone, blunt body, or Euler solver) as well as a number of parameters denoting the status of the run.

```

D:\Users\dfrobins\Zeus\Distribution>echo off
7 File(s) copied
7 File(s) copied

**** (Run0001) **** Free-Stream Starting Solution ****
**** (Run0001) **** Executing ZEUS (Free-Stream Inflow Profile)****

STEP = 25  ZETA = 3.500E-02  DZETA = 1.000E-03  NCFL.MCFL = 1  18
STEP = 50  ZETA = 6.000E-02  DZETA = 1.000E-03  NCFL.MCFL = 1  6
STEP = 75  ZETA = 8.500E-02  DZETA = 1.000E-03  NCFL.MCFL = 1  19
NORMAL STOP

**** (Run0001) **** Run Completed ****

Press any key to continue . . .

```

FIGURE 32. EULER CONSOLE OUTPUT

The output written to the command prompt for the inviscid solver includes the following parameters:

- *Starting profile* - Denotes which type of starting solution is employed (freestream, cone, or blunt body). This parameter is specified in the Integration Control section.
- *Step* - Counter for the number of axial steps taken. The frequency with which the screen data is written is controlled by the Output Residual parameter in the Output Control section.
- *Zeta* - Axial location of the current integration step.
- *dZeta* - Local axial stepsize. This value changes as the solution proceeds downstream and is controlled by the stability criterion specified in the Integration Control section. If the axial stepsize is too large to resolve the geometric features then the maximum allowable stepsize can be adjusted in the Integration control section.
- *nCFL, mCFL* - The radial and circumferential index for the maximum residual.
- *Exit Status* - Upon termination the exit status of the current run is displayed along with any error messages that occurred during the Euler solution. 'Normal Stop' is displayed if no runtime errors were encountered.

Note: After the run has completed the window will display 'Press Any Key' to continue. When a key is depressed, the command window is deleted and control is returned to the Zeus<sup>++</sup> graphical user interface

window. Whether or not the window pauses after completion is controlled by the 'Pause After Each Run' parameter in the Options section.

### 5.1.2 Boundary Layer Console Output

```

C:\WINNT\System32\CMD.exe

D:\Users\dfrobins\Zeus\Distribution>echo off
7 File(s) copied
7 File(s) copied

xxxx (Run0001) xxxx Executing Boundary Layer Code (Adiabatic) xxxx
      Step =      100      Zeta =      6.4860E-02
      Step =      200      Zeta =      1.1972E-01
      Step =      300      Zeta =      1.7458E-01
      Step =      400      Zeta =      2.2944E-01
      Step =      500      Zeta =      2.8430E-01
      Step =      600      Zeta =      3.3916E-01
      Step =      700      Zeta =      3.9402E-01
      Step =      800      Zeta =      4.4888E-01
      Step =      900      Zeta =      5.0374E-01
Normal Stop

```

FIGURE 33. BOUNDARY LAYER CONSOLE OUTPUT

The output written to the command prompt for the viscous boundary layer solver includes the following parameters:

- *Case description* - During each run the case number being executed is displayed (Runxxxx) along with the appropriate heat flux boundary condition at the surface (i.e., Adiabatic or Isothermal). The boundary condition is specified in the General Aerodynamic Data section.
- *Step* - Counter for the number of axial steps taken. The frequency with which the screen data is written is controlled by the Output Residual parameter in the Output Control section.
- *Zeta* - Axial location of the current integration step.
- *Exit Status* - Upon termination the exit status of the current run is displayed along with any error messages that occurred during the Euler solution. 'Normal Stop' is displayed if no runtime errors were encountered.

Note: After the run has completed the window will display 'Press Any Key' to continue. When a key is depressed, the command window is deleted and control is returned to the Zeus<sup>++</sup> graphical user interface window. Whether or not the window pauses after completion is controlled by the 'Pause After Each Run' parameter in the Options section.

## 5.2 ERROR MESSAGES

### Overview

The possible error messages encountered while executing the Zeus<sup>++</sup> code are listed below for each section of the tool. Note that 5000 series errors occur in the Euler solver, 6000 series in the boundary layer solver, 7000 series in the cone solver, and 8000 series in the blunt body solver. A description for each series is listed in the corresponding section below.

<u>Zeus Code</u>	Error messages for the Zeus Euler code (inviscid solution).
<u>Boundary Layer Code</u>	Error messages for the Boundary Layer code (viscous solution).
<u>Cone Code</u>	Error messages for the Cone code (generates starting profiles).
<u>Blunt Body Code</u>	Error messages for the Blunt Body code (generates starting profiles).

### 5.2.1 Zeus Error Messages

<u>5101 - 5104</u>	Maximum iteration count to locate corner #1-#4 exceeded.
<u>5105 - 5106</u>	Negative pressure encountered while decoding conservative variables.
<u>5108</u>	Undefined surface geometry parameter.
<u>5109</u>	Forward swept leading edges (control surfaces) not yet implemented.
<u>5110</u>	Rearward swept trailing edges (control surfaces) not yet implemented.
<u>5111 - 5112</u>	After first order fix the flow remains subsonic.
<u>5115 - 5116</u>	Negative square root occurred in the Prandtl-Mayer expansion calculation.
<u>5118</u>	Error in linear interpolation for shockwave.
<u>5120</u>	Dimensions are too large.
<u>5121</u>	MAZ() array not consistent with MA() array.
<u>5123</u>	Specified radial clustering not attainable for aft GASP grid.
<u>5124</u>	Undefined control surface input.
<u>5128 - 5129</u>	Clustering parameters not consistent between the Cone code and the Zeus code.
<u>5130</u>	Error in oblique shock calculation.
<u>5131</u>	No possible oblique shock solution.
<u>5132</u>	No convergence in Prandtl-Meyer calculation.
<u>5134</u>	The number of specified axial plotting stations is greater than the maximum allowed. The MaxPlots parameter must be increased in the parameter statement and the code must be recompiled.
<u>5135</u>	Invalid starting solution passed to the Euler solver. Typically caused by errors in the cone or blunt body starting profile generators.

### 5.2.2 Boundary Layer Error Messages

<u>6001</u>	Maximum iteration count reached
<u>6002</u>	Invalid axial force calculation

5.2.3 Cone Error Messages

7001            Shock angle too great. If excessive turning angles are encountered during the cone solution then the initial profile will be subsonic.

5.2.4 Blunt Body Error Messages

8001            Errors in the blunt body starting profile generator.

## CHAPTER 6: POST-PROCESSOR

### OVERVIEW

The Zeus<sup>++</sup> tool includes a built-in post-processor for viewing the solution data. A variety of methods of examining the results are available including ASCII representations of the data as well as plots of a wide variety of parameters, all of which are discussed in detail below. The plotting features of the Zeus<sup>++</sup> code make use of the Tecplot<sup>9</sup> plotting package.

Note: Zeus<sup>++</sup> can be used if Tecplot is not installed on the users' machine, however, the plotting capabilities of the post-processor will not be functional. ASCII representations of the data will still be available.

The 'PostProcessor' menu option offers the following commands:

#### ASCII Forces/Moments

Examine an ASCII data file of the forces/moments and centers of pressure acting on the body.

#### Single Run Plots

Line plots of the surface pressures, forces, and moments. Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc.).

#### Multi-Run Plots

Comparative line plots of forces/moments versus a variety of independent variables (Mach number, angle of attack, yaw angle, and roll orientation).

### 6.1 ASCII Data

#### 6.1.1 Forces & Moments

There are two options when examining the forces, moments, and centers of pressure for a given set of runs. The first is to view the totals or the sum from each of the zone edges. This gives the sum of the values from the missile surface and the control surfaces, or the total acting on the geometry. The second option prints the forces/moments separately on each edge of each zone. This is often useful if the values on a particular control surface are desired (i.e., force due to 1 fin). Recall, that edge<sub>1</sub> refers to the missile surface, edge<sub>2</sub> and edge<sub>3</sub> refer to the zonal boundaries (i.e., the control surfaces, if they exist), and edge<sub>4</sub> refers to the outer boundary (see figure below).

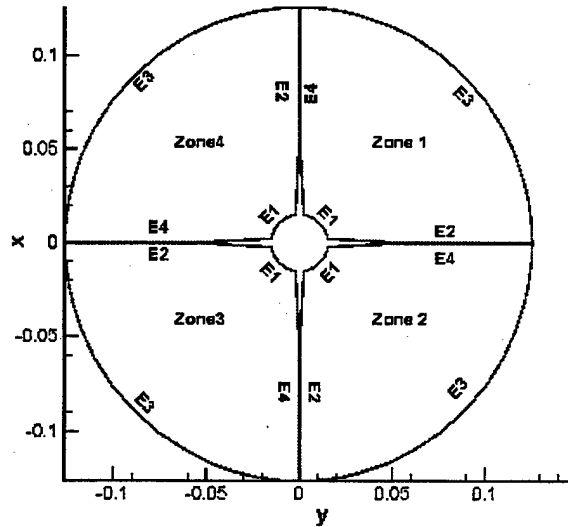


FIGURE 34. EULER GRID REFERENCE DIAGRAM

For each case run, the ASCII output file, 'ForcesMoments.txt' will contain the following data:

Mach #  
 Angle of Attack  
 Angle of Yaw  
 Roll Angle  
 Normal, Side, and Axial Force  
 Yaw, Pitch, and Roll Moments  
 $X_{cp}/L_{Ref}$  and  $Y_{cp}/L_{Ref}$  (centers of pressure)

Note: An approximate value for the axial force due to the base (i.e., base drag) is calculated using the methodology of reference 14. Note that the corrections due to angle of attack and fin thickness have been omitted. Also, note that the base drag calculation requires the ratio of the base area to the reference area ( $A_{base}/A_{ref}$ ). Zeus<sup>++</sup> calculates the base area assuming a circular cross-section, and therefore, if a non-circular cross-sectional geometry is applied, the resultant base drag must be multiplied by the factor ( $A_{base}/A_{circ}$ ). In other words, an equivalent circular base area must be applied.

If both the inviscid and viscous solutions have been performed then the output for the forces/moments will be listed componentwise. In other words, the inviscid, viscous, and induced contributions will be listed along with the totals.

## 6.2 SINGLE RUN PLOTS

### Overview

This section of the post-processor is used to examine the output data for a single run. To make a comparative analysis of different runs see the Multi-Run Plots section.

The 'single Run Plots' menu offers the following three choices:

#### Surface Pressures

Line plot of the surface pressures.

#### Forces and Moments

Line plots of the forces, moments, and centers of pressure.

#### Contour/Vector Plots

Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc.).

### 6.2.1 Surface Pressures

The first option, 'Plot Surface Pressure', will display a graph of the surface pressure versus axial distance for zone edges 1, 2, and 4 (see figure in ASCII data section). The pressures are listed for each circumferential plane on the missile surface and for each radial plane on the zone edges (i.e., control surfaces). Each plane is labeled as 'Plane #x, Zone #y, Edge #z'. If the Edge number is '1' then the Plane # refers to the circumferential index. If the Edge number is '2' or '4' then the Plane # refers to the radial index.

### 6.2.2 Forces and Moments

The second single-run plotting option, 'Plot Forces & Moments', will plot either the total force/moment or the edge force/moment as a function of axial location. The totals versus edge values are described in the ASCII data section. The 'x', 'y', 'z' coordinates refer to normal, side, and axial components, respectively.

### 6.2.3 Contour/Vector Plots

This plotting option generates contour plots of a variety of flowfield variables for a given run. Click on 'Choose Plot Attributes' to bring up the dialog box shown below and specify the plotting attributes. Select the dimensions and the variables desired for the contour plot. Note that the coordinates (x, y, z) and the velocities (u, v, w) are always written to the contour plot file. After the plot information has been specified, click 'OK' to accept the values. Then select 'Plot Data' to generate the contour plots.

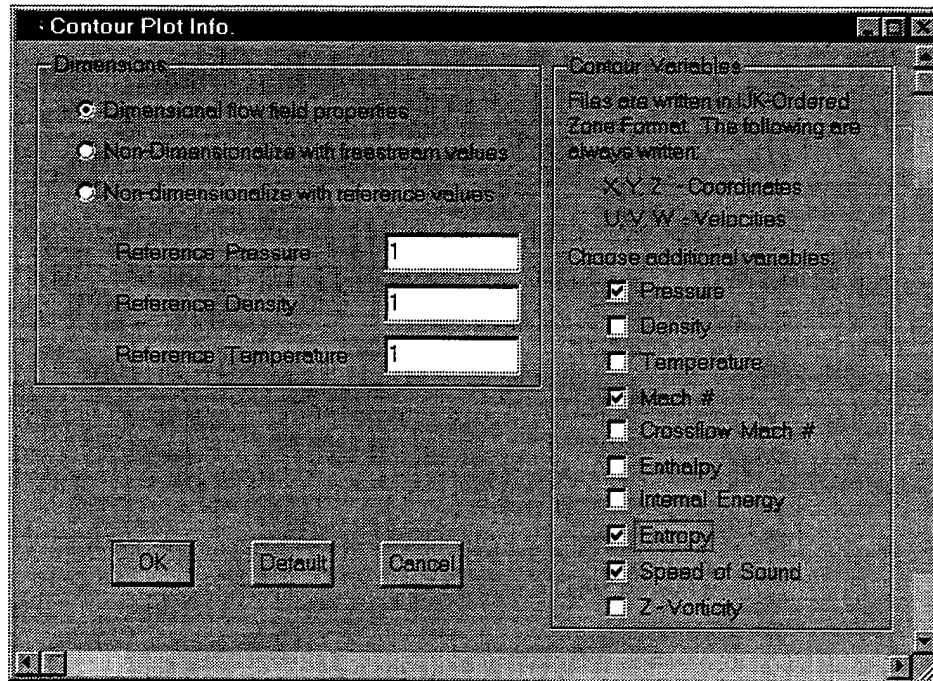


FIGURE 35. CONTOUR PLOT CONTROLS

### 6.3 MULTI-RUN PLOTS

#### Overview

This section of the post-processor is used to compare the effect of Mach number, angle of attack, yaw angle, and roll orientation on the forces, moments, and centers of pressure of a given geometry. Select the desired independent variable as well as the desired dependent variables (see figure below). Click on 'OK' to generate a plot for all of the completed runs.

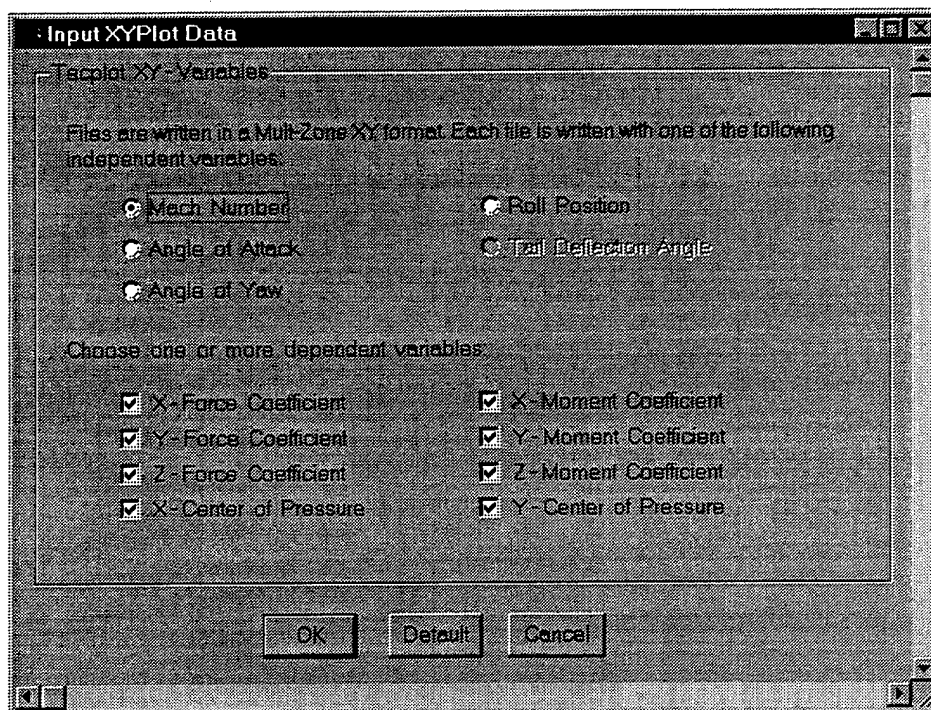


FIGURE 36. 24 PLOT CONTROLS

## 7.0 REFERENCES

1. GASP v3 Computational Fluid Dynamics Solver, Aerosoft, Inc., 1998.
2. Wardlaw, A.B.; Davis, S.F.; and Priolo, F.J., *A Second Order Gudonuv's Method for Supersonic Tactical Missile Computations*, NSWC TR 86-506, Dec 1998.
3. Hsieh, T. and Priolo, F.J., *Generation of the Starting Plane Flowfield for Supersonic Flow Over a Spherically Capped Body*, NSWC TR 84-484, May 1985.
4. Wardlaw, A.B. and Priolo, F.J., *Applying the Zeus Code*, NSWC TR-86-508, Dec 1986.
5. Mendenhall, Michael R., "Tactical Missile Aerodynamics: Prediction Methodology," Vol. 142, Progress in Astronautics and Aeronautics, 1992.
6. Wardlaw, A.B. and Baltakis, F.P., "An Integral Boundary Layer Procedure for Tactical Missiles," AIAA 92-1026, Aerospace Design Conference, Irvine, CA, Feb. 1992.
7. Wardlaw, A.B.; Priolo, F.J.; and Solomon, J.M., *A Multiple Zone Method For Supersonic Tactical Missiles*, NSWC TR 85-484, Jun 1986.
8. Wardlaw, A.B. and Priolo, F.J., Personal Communication and unpublished documentation.
9. Tecplot v7.0 Plotting Package, Amtec Engineering, 1998.
10. Hymer, T.C.; Moore, F.G.; and Downs, C., *User's Guide for an Interactive Personal Computer Interface for the Aeroprediction Code*, NSWC-DD TR 94-107, Jun 1994.
11. Baltakis, F.P.; Wardlaw, A.B.; Jr., Allen, J.M., "Leeside Crossflow Modeling in Euler Space Marching Computations," NSTC TR 86-342, Nov 1986.
12. Kwong, C.M. and Myring, D.F., "Fusiform Body Separation Flow Field Calculations using Euler and Boundary Layer Methods", Royal Aeronautics Society Conference on the Prediction and Exploitation of Separated Flows, Paper No. 27, Apr 1989.
13. Monta, J. W., "Supersonic Aerodynamic Characteristics of a Sparrow III Type Missile Model With Wing Controls and Comparison With Existing Tail-Control Results," NASA Technical Paper #1078, 1977.
14. Moore, F.G. and Hymer, T., "Improved Empirical Model for Base Drag Prediction on Missile Configurations Based on New Wind Tunnel Data," NSWC-DD TR 92-509, Oct 1992.
15. Shapiro, A.H., "The Dynamics and Thermodynamics of Compressible Fluid Flow," Vol. I, pp. 211-212, 1953.

**APPENDIX A**  
**TUTORIAL**

## A.1 INTRODUCTION

### STARTING ZEUS++

Execution of the code begins by double clicking on the program 'ZeusPP.exe', entering a name for the current case ('tutorial'), and then clicking on 'start New Case'. For the tutorial, we will be considering a SeaSparrow missile<sup>5,13</sup> with both dorsal and tailfins.

Note: The name entered cannot contain blank spaces.

Execution of the Zeus++ code can be divided into four basic sections:

1. Geometry Setup
2. Case Inputs
3. Code Execution
4. Post-Processing

## A.2 GEOMETRY SETUP

### OVERVIEW

The geometry setup section can be sub-divided into four areas:

- 1) Specify the grid parameters
- 2) Define the missile geometry
- 3) Define the control surfaces
- 4) Generate the computational domain (optional).

#### A.2.1 GRID PARAMETERS

The properties that define the computational grid are specified in this section. Each of these parameters is described in detail in the Grid Parameters section.

- From the main menu, click on 'Geometry' and then select 'Grid Parameters' and 'Euler Grid Properties'. For an explanation of each of the parameters on this screen, press the 'F1' key. The default values are adequate for the current tutorial, therefore, simply click on 'OK' to accept the current values.

Note: From the 'Grid Parameters' option, it is seen that there are two sets of grid parameters available. In the first (Euler Grid Properties), the parameters specified are applied during both the Zeus Euler solution as well as the viscous boundary layer solution. In the second (Navier-Stokes Grid Properties), the parameters specified are used to generate a grid, which can be exported to a full Navier-Stokes computational fluid dynamics solver. The Navier-Stokes parameters are used only for grid generation purposes, and therefore, if the user does not intend to export the grid to an N-S solver, this section may be omitted.

### A.2.2 MISSILE SECTIONS

After specifying the grid parameters, the next step is to define the missile geometry. For the tutorial, the missile geometry will be entered as three separate sections: 1) Tangent-Ogive nose, 2) cylindrical mid-body and 3) boat-tail flare.

The first section is the Tangent-Ogive Nose

- Click on 'Geometry' and then select 'Missile Sections'.
- Click on the down arrow next to 'select Geometry Type' and select 'Tangent Ogive'.
- Click on 'Add New Section'.
- Enter the final axial location (0.0686) of the nose as well as the final radius (.01524). Note that the missile starts at 0,0.
- Click on 'OK' and the code will calculate and display the initial missile section.

The next section is the cylindrical mid-body and is described by a conic section.

- Click on the down arrow next to 'select Geometry Type' and then select 'Conic Section'.
- Click on 'Add New Section'.
- The initial axial location and radius are copied from the end of the previous section.
- Enter the final axial location (0.537) of the conic as well as the final radius (.01524).
- Click on 'OK'.

The final section is the boat-tail flare and is described by a conic section.

- Click on the down arrow next to 'select Geometry Type' and then select 'Conic Section'.

- Click on 'Add New Section'.
- The initial axial location and radius are copied from the end of the previous section.
- Enter the final axial location (0.5486) of the conic as well as the final radius (.013715).
- Click on 'OK'.
- Click on 'Finished'
- Press 'Ctrl-R' to regenerate the geometry.

### A.2.3 CONTROL SURFACES

After specifying the missile geometry, the next step is to define the control surfaces. For the tutorial, two sets of control surfaces are considered. The first is the dorsal fins and the second is the tail fins. The control surfaces must be entered in the order in which they appear on the missile, therefore, the dorsal fins must be entered first, followed by the tail fins.

The first set of control surfaces (dorsal fins) will be generated using the 'Modified Wedge' option.

- Click on 'Geometry' and then select 'Control Surfaces'.
- Click on the arrow next to 'select Control Surface' and select 'Modified Wedge'.
- Click on 'Add Control Surface'.
- Enter a label for the control surface ("Dorsal Fins").
- Enter the axial location of the control surface hinge line ("0.2743"). The hinge line is the point on the control surface about which deflections are made.
- Enter the offset angles (in degrees) for each of the fins. For the current case the fins are located in the '+' configuration, and therefore, the offset angles are 90, 180, 270, and 360°.

Note: It is possible to delete an individual fin by right clicking on the fin you wish to remove. Both the offset angle and the deflection angle will be replaced with '——' to indicate that they have been eliminated. In order to add the fin back into the calculation, simply right click on it a second time. This feature is useful when you want to have a different number of fins for each set of control surfaces (i.e., 2 dorsal fins and 4 tail fins).

- Enter the deflection angles (in degrees) for each of the fins. Use +5.0 for the 90 and 270° fins.

Note: The fins are always deflected in a clockwise orientation (i.e., entering +5 for both fins will generate an asymmetric deflection).

- Click 'OK'

- The parameters used to describe the modified wedge control surface are described in detail in the Modified Wedge section. For the Dorsal fins, enter the following values and click on 'OK' when completed:

Modified Wedge Control Surface		
Enter the leading edge half angle	3.018	[deg]
Enter the trailing edge half angle	3.977	[deg]
Enter the leading edge sweep angle	45	[deg]
Enter the span (root to tip distance)	.0597	
Distance from leading edge to hinge line	.0422	
	Root	Tip
Enter the leading wedge thickness	.002225	3.822e-4
Enter the trailing wedge thickness	.002225	5.04e-4
Enter the chord	.0742	.0145
<input type="button" value="OK"/> <input type="button" value="Cancel"/>		
<input type="button" value="Convert to 'User Specified Geometry'"/>		

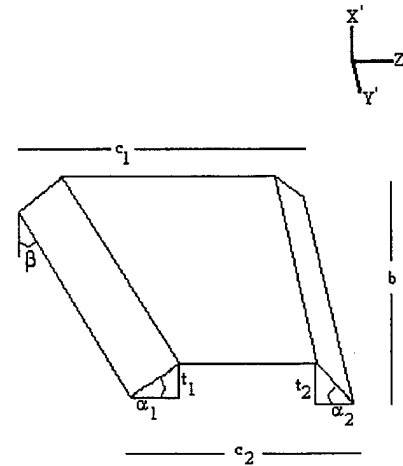


FIGURE A-1. MODIFIED WEDGE SAMPLE INPUT

The tailfins represent the second set of control surfaces. Because of the layout of the geometry of these fins, they will be generated using the 'User Defined Geometry' option.

- Click on the arrow next to 'select Control Surface' and select 'User Defined Geometry'.
- Click on 'Add New Section'.
- Enter a label for the control surface ("Tail Fins").
- Enter the axial location of the control surface hinge line ("0.5167").
- Enter the offset angles (in degrees) for each of the fins. For the current case the fins are located in the '+' configuration, and therefore, the offset angles are 90, 180, 270, and 360°.
- Enter the deflection angles (in degrees) for each fin (0°)
- Click 'OK'
- To ensure that the tail fins are described by planar surfaces the geometry must be entered as three cross sectional cuts. A detailed overview of the methodology used to generate this control surface geometry can be found at User Defined Geometry. For the tail fins under consideration, enter the following values for the three cross-sectional planes:

**Control Surface Definition**

(Plane #1 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e.  $z=0$  is the axial location of the hinge line)

'X' points

'Y' points

'Z' points

(Use spaces between points)

FIGURE A-2. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 1)

- Click 'Next Plane'

**Control Surface Definition**

(Plane #2 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e.  $z=0$  is the axial location of the hinge line)

'X' points

'Y' points

'Z' points

(Use spaces between points)

FIGURE A-3. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 2)

- Click 'Next Plane'

**Control Surface Definition**

(Plane #3 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e.  $z=0$  is the axial location of the hinge line)

'X' points: 0.00000000

'Y' points: .047725 .047725 .047725 .047725

'Z' points: .0319 .0319 .0319 .0319

(Use spaces between points)

Next Plane    Delete Plane    Finished

FIGURE A-4. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 3)

- Click 'Finished'
- Click 'Finished'
- Press 'Ctrl-R' to regenerate the geometry

#### A.2.4 GRID GENERATION (OPTIONAL)

- View Euler Grid

The purpose of this section is to examine the inviscid grid that will be used by the Zeus Euler solver to ensure that an acceptable grid is being employed.

Note: 1) The axial step size for the inviscid solution is calculated at runtime, however, the axial step size for grid generation purposes is taken to be constant.

2) The outer boundary, for grid generation purposes, is taken to be that specified in the Euler grid parameter section.

- Click on 'Geometry' and then select 'Grid Generation' and 'View Euler Grid'
- Click 'Plot Grid as a Series of 2d planes'
- Enter a starting axial location (0.01)
- Enter a final axial location (0.5486)
- Enter the number of axial step (25)
- Click 'OK'
- After the Tecplot window appears:
- Click 'File', 'Macro', and then 'Play'
- Double click on the '2dGrid\_Movie.mcr' file in the 'Playback' directory.

Note: The macro file will step axially down the missile and take a snapshot of each location. The viewport shown during the movie generation (i.e., x and y-axis ranges) can be specified in Zeus++ before generating the Euler grid by clicking on 'View' and then selecting 'Options'. Enter the desired values for the 'x' and 'y' axis ranges. A detailed description of all of the parameters is provided in the View/Options section.

- A movie of the grid will be created and written out as 'Playback\Zeus\_grid2d.rm'. This movie file can be replayed using the Tecplot Framer program.
- Navier-Stokes Grid

The Zeus++ code functions both as an inviscid solver (with approximate viscous corrections), and a CFD grid generator. This allows one to export a three-dimensional computational grid and the appropriate boundary conditions to a full Navier-Stokes solver. Currently, the grid and boundary condition files generated are suitable for the GASP flow solver. One circumferential plane of the Seasparrow Navier-Stokes grid is shown below with an explanation of each parameter provided in the Navier-Stokes Grid section.

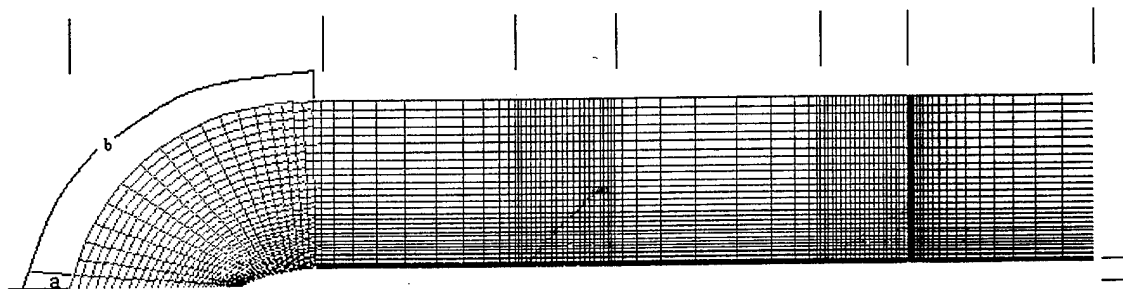


FIGURE A-5. SAMPLE OUTPUT OF NAVIER-STOKES GRID GENERATOR  
(2d CROSS-SECTION SHOWN)

- Click on 'Geometry', 'Grid Generation', 'Navier-Stokes Grid', and 'Generate Grid'
- Enter the number of axial sections (6). The geometry is separated into a number of axial stations so that clustering can be performed at a number of axial locations. From the above figure, the six axial sections are shown between the vertical lines. The six sections are as follows:

1. Nose section
2. Section between nose and first set of fins
3. Section covering dorsal fins
4. Section between dorsal fins and tail fins
5. Section covering the tail fins
6. Section aft of the missile

Note: If you extend the grid beyond the aft end of the missile (shown in blue between the vertical lines), then an additional section will be included to calculate the flow directly behind the missile (shown in yellow between the horizontal lines).

- Enter the number of axial stations, ending axial location, beginning spacing, and ending spacing for each of the six sections (values provided below):

Note: To simplify the initial estimate for axial spacing a 'Reset Spacing' button is provided. Simply enter the number of axial stations and the ending axial location for each of the desired sections. Then, click on 'Reset Spacing' to obtain an equally spaced axial grid. This grid can then be modified to provide the desired axial clustering.

# of 'Z' stations	Ending 'Z' location	Beginning Spacing	Ending Spacing
20	.0686	.003	.005
12	.230	.01	.003
20	.3064	.003	.003
12	.4775	.003	.003
21	.5486	.003	.003
20	.7	.001	-NA-

**Aft Grid**  
☒ Include grid aft of missile  
 Enter the # of radial pts. (centerline to surface)

**Outer Boundary Clustering**  
 Centerline to first point angle  [deg]  
 Centerline to last point angle  [deg]

OK Cancel

FIGURE A-6. SAMPLE INPUTS FOR NAVIER-STOKES GRID GENERATOR

- For the aft grid section, select the 'Include Grid Aft of Missile' and enter the number of radial points to include (15).
- In order to control the clustering of the outer boundary of the nose, the beginning and ending angles (see 'a' and 'b' in the above figure) of the outer boundary are specified. Enter the values shown in the above figure.
- Click on 'Ok' to bring up the boundary condition window (see Figure A-7).
- Select the surface boundary condition.
- Select the two-equation turbulence model.
- Click on 'OK' to generate the computational grid and boundary condition files.

Note: The output from the grid generation code is placed in the 'filename/RunGrid' directory. For the current case, this is 'Tutorial/RunGrid'. The two files are the grid geometry in Plot3d format (GASPGGrid.p3da) and the GASP boundary condition input file (GASPbc.inp).

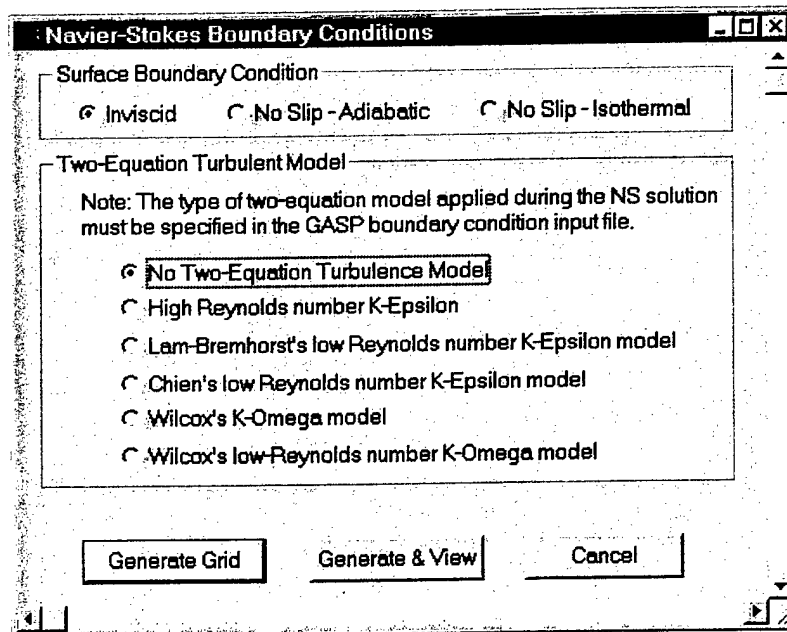


FIGURE A-7. SAMPLE BOUNDARY CONDITION INPUTS FOR NAVIER-STOKES GRID GENERATOR

### A.3 INPUT PARAMETERS

#### OVERVIEW

After the computational domain, missile, and control surface geometries have been specified, the next step is to input the following parameters:

1. Aerodynamic Data
2. Integration Controls
3. Separation Modeling Parameters
4. Boundary Layer Controls
5. Output Controls

### A.3.1 AERODYNAMIC DATA

This section is used to specify the Mach numbers and missile orientations (angle of attack, yaw, and roll) to be used during execution of the flow solver. Rather than specifying singular values, sweeps are defined for each of the above mentioned parameters. In order to generate a given sweep, input the beginning, ending, and incremental value for the desired parameter (see below).

- Click on 'Inputs' and then select 'Aerodynamic data'.
- Select the 'Aerodynamic Run Matrix' tab and then enter the values shown in the figure below.

The screenshot shows a dialog box titled "Aerodynamic Run Matrix" with a tab labeled "General Conditions". The dialog is divided into several sections for defining sweeps:

- Mach Number Sweep:**
  - First Mach Number: 2
  - Last Mach Number: 4.5
  - Mach Number Intervals: 0.25
- Configuration Roll Position Sweep:**
  - First Roll Position: 0
  - Last Roll Position: 45
  - Roll Position Intervals: 22.5
- Angle of Attack Sweep:**
  - First Angle of Attack: 0
  - Last Angle of Attack: 20
  - Angle of Attack Intervals: 10
- Deflection Sweep:**
  - First Tail Deflection: 0
  - Last Tail Deflection: 1
  - Tail Deflection Intervals: 1
- Angle of Yaw Sweep:**
  - First Angle of Yaw: 0
  - Last Angle of Yaw: 4
  - Angle of Yaw Intervals: 4

At the bottom right, there is a button labeled "Default Values". At the bottom of the dialog are four buttons: "OK", "Cancel", "Apply", and "Help".

FIGURE A-8. SAMPLE OF AERODYNAMIC RUN MATRIX PARAMETERS

The next section is used to specify the remaining aerodynamic parameters including free-stream atmospheric conditions, units, reference conditions (diameter, area, and moment arm), viscous parameters (wall heat flux boundary condition, Prandtl number), as well as the gas type (real or ideal gas).

- Click on the 'General Conditions' tab to specify the remaining aerodynamic parameters.

- The free-stream aerodynamic parameters (density, pressure, and ratio of specific heats) can be specified in a variety of ways. For the current example, a unit Reynolds number is provided and the free-stream quantities are computed. Alternate methods include explicitly specifying the values or specifying an altitude and using standard atmospheric tables to look up the corresponding values.
- Specify the units as metric.
- Enter the reference diameter, area, and moment arm shown in the figure below.
- Select 'Adiabatic' and specify a Prandtl number of 0.72.

Note: The viscous parameters are not used in the Zeus Euler solver. These values are, however, required inputs for the approximate viscous (boundary layer) solver. Therefore, these values are not required unless you are running the viscous solution along with the inviscid flow solver.

- Click on 'OK' and wait for the code to generate the required run matrix.

**Aerodynamic Run Matrix - General Conditions**

**Freestream Data**

☐ 1962 U.S. Standard Tables

☐ 1959 ARDC Tables

Altitude  [ft, m]

☐ Pressure

☐ Density

☐ Gamma

☒ Unit Reynolds Number

**Units**

☐ English ( $p=\text{lb}/\text{ft}^2$ ,  $\rho=\text{slug}/\text{ft}^3$ ,  $T=R$ ,  $xyz=\text{ft}$ )

☒ Metric ( $p=\text{N}/\text{m}^2$ ,  $\rho=\text{kg}/\text{m}^3$ ,  $T=K$ ,  $xyz=\text{m}$ )

**Reference Conditions**

Reference Diameter

Reference Area

Axial Moment Reference

**Viscous Parameters**

☒ Adiabatic

☐ Wall temperature

Prandtl number

**Gas Type**

☒ Perfect Gas ☐ Real Gas

OK Cancel Apply Help

FIGURE A-9. SAMPLE INPUTS FOR GENERAL AERODYNAMIC PARAMETERS

### A.3.2 INTEGRATION CONTROL

This section is used to specify integration parameters, all of which are described in detail in the Integration Parameters section.

- Click on 'Inputs' and then select 'Integration Control'
- Enter all of the values shown in the figure below and click 'OK'.

FIGURE A-10. SAMPLE INPUT FOR INTEGRATION PARAMETERS

### A.3.3 SEPARATION MODELING

This section is used to specify separation-modeling parameters, all of which are described in detail in the Separation Modeling section. Separation modeling is not used for the tutorial, and therefore, this section may be omitted.

### A.3.4 BOUNDARY LAYER DATA

If an approximate viscous solution is desired then the following values must be specified. An approximate viscous solution is obtained by solving the integral form of the boundary layer equations. The integral boundary layer solution employed requires the inviscid

solution from the Zeus Euler solver as a boundary condition. An in-depth description of all of the parameters shown below are described in the Boundary Layer Parameters section.

- Click on 'Inputs' and then select 'Boundary Layer Data'.
- Enter the values shown in the figure below and click on the 'Modeling Parameters' tab.

Execution Parameters | Modeling Parameters | Output Parameters

Boundary Layer Parameters

Output inviscid flowfield every 'iplotbl' axial steps 1000

Output inviscid flowfield if axial stepsize > 'delbl' 999.9

Integration Control

Starting axial location for integration 0.02

Minimum number of axial steps 500

Maximum number of axial steps 40000

Number of implicit iterations 10

Method for calculating effective radius

☒ Streamline ☐ Geometry ☐ None

OK Cancel Apply Help

FIGURE A-11. SAMPLE INPUTS FOR BOUNDARY LAYER EXECUTION PARAMETERS

- Enter the values shown in the figure below and click on 'OK'

Execution Parameters | **Modeling Parameters** | Output Parameters

Transition Parameters

☐ Laminar

☒ Axial transition location

☐ Gradual transition factor

Modeling Factors

Minimum pressure gradient to initiate shock separation

Streamline divergence factor

Streamline convergence factor

Induced pressure limiter

Momentum and energy thickness limiter

OK Cancel Apply Help

FIGURE A-12. SAMPLE INPUTS FOR BOUNDARY  
LAYER MODELING PARAMETERS

### A.3.5 OUTPUT CONTROLS

This section is used to specify what type of output is generated, as well as the frequency of the output (with respect to axial integration steps). Each of the parameters shown below are described in detail in the Output Controls section.

- Click on 'Inputs' and then select 'Output Control'
- Enter all of the values shown in the figure below and click 'OK'.

**Output Controls**

Surface Pressures - Forces - Moments

☐ Print cell coordinates  
☒ Do not print cell coordinates

Write surface properties at every 'k' axial station: 50

Write surface properties if stepsize is greater than DELZA: 5000

---

Contour Data

Write contour properties at every 'k' axial station: 200

and

at the following user specified axial locations: 0.20 0.22 0.23 50 51 52  
(Must be in ascending order)

Write contour properties if axial stepsize is greater than DELZC: 5000

---

ASCII Data

Output residual every ISKIP iterations: 10

Print crossflow plane if step number is evenly divisible by IPRINT: 5000

Print N planes which are evenly divisible by NSKIP: 1

Print M planes which are evenly divisible by MSKIP: 1

OK Cancel

FIGURE A-13. SAMPLE INPUTS FOR OUTPUT CONTROL SECTION

## A.4 EXECUTION

### RUNNING A CASE

This section is used to select which cases are evaluated as well as what type of solution is performed.

- Click 'Execution'.
- Click on the desired run.

Note: Multiple runs may be selected in a number of ways. The first is to hold down the 'Ctrl' key and click on each of the desired runs. Alternatively, you can click on the first desired run, hold down the 'shift' key, and then click on the last run to select all cases in between. Finally, you can simply click on the 'select All' button to run all cases.

- Select the type of run desired.

Note: If 'Full Run' is selected the code will execute the Inviscid Euler solver, write out the flowfield at the specified axial intervals, and then execute the boundary layer solver to get an approximate viscous solution. If 'Inviscid Run' is selected, the code will execute the Inviscid Euler solver and then terminate. No viscous approximation will be calculated. If a viscous calculation is desired at later time, simply click on the 'Viscous Run' button, and select the desired case. Only those runs that have been solved with the inviscid solver will be displayed when the 'Viscous Run' button is selected. This is because the viscous approximation requires the inviscid flowfield as a boundary condition.

- Specify whether a symmetry plane exists.

Note: If you know a priori that the geometry is non-symmetric about the pitch plane, you must select 'Full Grid (All Runs)'. However, if the geometry is symmetric about the pitch plane choose either 'symmetry Plane (All Runs)' if you will not be running any asymmetric cases, or alternatively, choose 'symmetry Plane (Zero Roll and Yaw)' to run a symmetry plane only for those cases with zero yaw or roll angles. Running with a symmetry plane will cut the computational time approximately in half.

- Click on 'OK' to perform the desired calculations.
- After the runs have completed, you can go back to the 'Execution' section to check on the termination status for each run. If the 'Completed' column contains one '\*', then the inviscid solution has completed. If it contains '\*\*' then both the inviscid and viscous solutions have been completed. In addition, the 'Exit Status' column should contain 'Normal Exit' if no problems were encountered. If the code did not exit normally then an exit code will be provided. The exit codes, along with an explanation of each, can be found in the Exit Codes section.

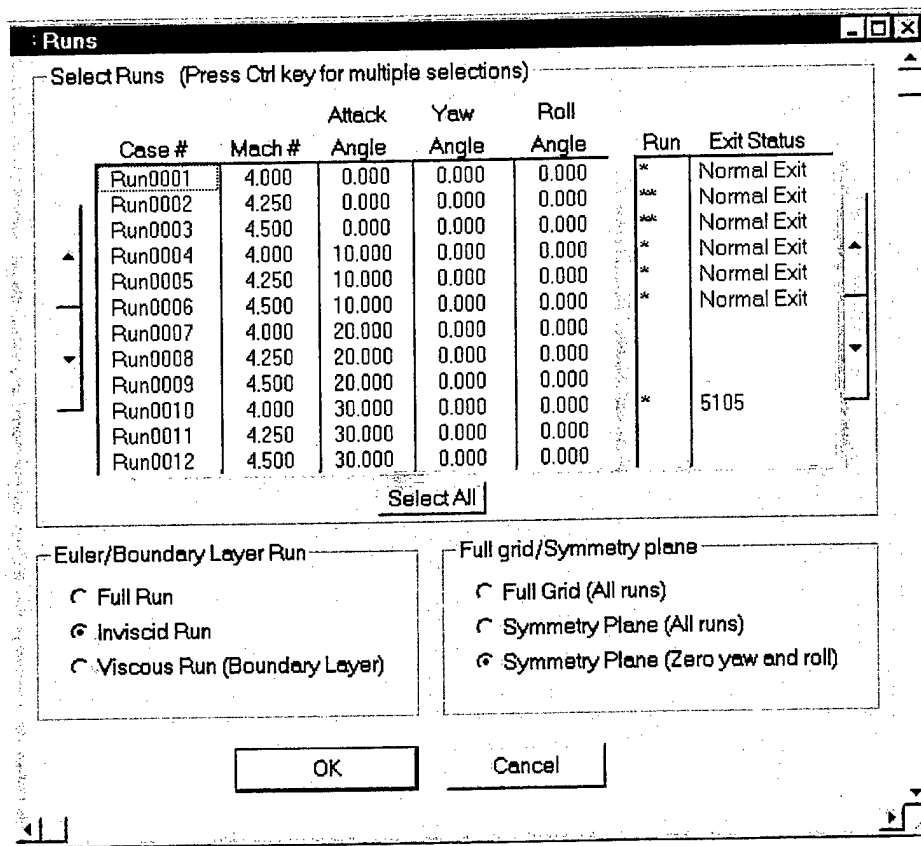


FIGURE A-14. SAMPLE SCREEN SHOT OF EXECUTION CONTROL PARAMETERS

## A.5 POST-PROCESSING

### OVERVIEW

The post-processing section of the tutorial is divided into three sections. To execute the plotting functions of the post-processor, a current version of the Tecplot<sup>9</sup> plotting software must be installed on the user's machine. If this software is not installed, the user may still examine the ASCII output data generated by Zeus<sup>++</sup>.

ASCII Data

Examine an ASCII data file of the forces/moments acting on the body as well as the centers of pressure.

Single Run Plots

Line plots of the surface pressures, forces, and moments. Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc).

Multi-Run Plots

Comparative line plots of forces/moments versus a variety of independent variables (Mach number, angle of attack, yaw angle, and roll orientation).

## A.5.1 ASCII FORCES &amp; MOMENTS

This section is used to examine the ASCII output data from the Zeus<sup>++</sup> runs. The data includes the forces, moments, and centers of pressure for both the entire geometry as well as each individual zone edge. An in-depth explanation of the ASCII data can be found in the ASCII Forces & Moments section of the users' manual.

- Click on 'Post-Processor' and then select 'ASCII Forces & Moments'
- Select 'Totals' to display the total forces/moments acting over the entire geometry. Alternatively, 'Edge Values' could be selected to display the individual components of the forces/moments that constitute the total.
- Microsoft WordPad will open and display the ASCII (forcesmoments.txt) version of the output data. The data can either be printed or saved to another file using this program. When finished examining the data, close the WordPad program to return to Zeus<sup>++</sup>.

## A.5.2 SINGLE-RUN PLOTS

This section is used to generate graphs (both line and contour) of the forces, moments, centers of pressure, and a variety of flowfield parameters (density, Mach number, pressure, etc...). Comparison plots of the various runs are generated in the Multi-Runs section.

- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Plot Surface Pressure'. A window will then be displayed which lists all completed runs. Select the desired run and click on 'OK' to generate the graph. A detailed description of the data in the surface pressure plot can be found in the users' manual in the Surface Pressure section.
- Exit the plotting package to return to Zeus<sup>++</sup>.
- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Plot Forces & Moments' and then choose either 'Totals' or 'Edge Values' to generate the corresponding graph. After the completed runs are displayed, select the desired run and click on 'OK' to generate the graph. A description of the data in the graph is provided in the users' manual in the Forces & section.

- Exit the plotting package to return to Zeus<sup>++</sup>.
- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Contour Plots' and then 'Choose Plot Attributes'. Enter the desired dimensionalization parameters for the contour plots, select the desired variables, and click on 'OK'. The above parameters are explained in detail in the Contour/Vector section of the user's manual.
- Return to the 'Contour Plots' menu, however, this time click on 'Plot Data' to generate the contour graph. After the completed runs are displayed, select the desired run and click on 'OK' to generate the graph.
- Exit the plotting package to return to Zeus<sup>++</sup>.

### A.5.3 MULTI-RUN PLOTS

This section is used to make comparison plots of different runs. The options include comparisons of all forces, moments, and centers of pressure with a choice of dependant variables (Mach number, angle of attack/yaw/roll).

- Click on 'Post-Processor' and then select 'Multi-Run Plots'.
- Select 'Choose Plot Attributes' and specify the values to be plotted as well as the independent variable. Refer to the Multi-Plot run section of the user's manual.
- Return to the 'Multi-Run Plots' menu, however, this time click on 'Plot Data' to generate the comparison plot for all executed runs. If, during the tutorial, only a single run was executed, then the line plot displayed will be a point. In order to make comparative plots of different runs, return to the execution section, and run additional cases.
- Exit the plotting package to return to Zeus<sup>++</sup>.

## DISTRIBUTION

	<u>Copies</u>		<u>Copies</u>
DOD ACTIVITIES (CONUS)		ATTN T C TAI	1
		M J MALIA	1
ATTN CODE 35 (ZIMET)	1	TECHNICAL LIBRARY	1
CODE 351 (SIEGEL)	1	COMMANDER	
CODE 351 (CHEW)	1	NSWC CARDEROCK DIVISION	
CODE 332FD (LEKOUDIS)	1	WASHINGTON DC 20034	
CHIEF OF NAVAL RESEARCH			
BALLSTON CENTRE TOWER ONE		ATTN R M HOWARD	1
800 NORTH QUINCY ST		TECHNICAL LIBRARY	1
ARLINGTON VA 22217-5660		SUPERINTENDENT	
		NAVAL POSTGRADUATE SCHOOL	
ATTN CODE 474T6OD (LOFTUS)	1	1 UNIVERSITY CIRCLE	
CODE 4732HOD (SMITH)	1	MONTEREY CA 93943-5001	
CODE 473COOD (PORTER)	1		
CODE 47311OD (HOUSH)	1	ATTN HEAD WEAPONS DEPT	1
CODE 47311OD (GLEASON)	1	HEAD SCIENCE DEPT	1
CODE 4722EOD (JETER)	1	SUPERINTENDENT	
TECHNICAL LIBRARY	1	UNITED STATES NAVAL ACADEMY	
COMMANDER		121 BLAKE RD	
NAVAL AIR WARFARE CENTER		ANNAPOLIS MD 21402-5000	
1 ADMINISTRATION CIRCLE			
CHINA LAKE CA 93555-6001		ATTN DIAG DT 4T (PAUL MURAD)	2
		DIRECTOR	
ATTN TECHNICAL LIBRARY	1	DEFENSE INTELLIGENCE AGENCY	
G RUDACILLE PMS 38012 7	1	WASHINGTON DC 20301	
COMMANDER			
NAVAL SEA SYSTEMS COMMAND		ATTN BRENT WAGGONER	1
2531 JEFFERSON DAVIS HWY		CODE 4072 BLDG 2540	
ARLINGTON VA 22242-5160		NAVAL WEAPONS SUPPORT CENTER	
		CRANE IN 47522-5000	
ATTN TECHNICAL LIBRARY	1		
COMMANDER		ATTN CODE 5252P (KRAUSE)	1
NAVAL AIR SYSTEMS COMMAND		TECHNICAL LIBRARY	1
47122 LILJENCRANTZ ROAD UNIT 7		COMMANDER	
PATUXENT RIVER MD 20670-5440		INDIAN HEAD DIVISION	
		NAVAL SURFACE WARFARE CENTER	
ATTN C KLEIN	1	101 STRAUSS AVE	
TECHNICAL LIBRARY	1	INDIAN HEAD MD 20640-5035	
COMMANDER			
NAVAL AIR WARFARE CENTER		ATTN TECHNICAL LIBRARY	1
WEAPONS DIVISION		COMMANDING GENERAL	
521 9 <sup>TH</sup> ST		MARINE CORPS COMBAT	
POINT MUGU CA 93042-5001		DEVELOPMENT COMMAND	
		2048 SOUTH ST	
		QUANTICO VA 22134-5129	

**DISTRIBUTION (Continued)**

	<u>Copies</u>		<u>Copies</u>
ATTN E SEARS	1	ATTN R PUHALLA JR	1
L E LIJEWSKI	1	W STUREK	1
C COTTRELL	1	C NIETUBICZ	1
TECHNICAL LIBRARY	1	A MIKHAIL	1
AFATL (ADLRA) (DLGC)	1	P PLOSTINS	1
EGLIN AFB FL 32542-5000		TECHNICAL LIBRARY	1
ATTN TECHNICAL LIBRARY	1	COMMANDING GENERAL	
USAF ACADEMY		BALLISTIC RESEARCH LABORATORY	
COLORADO SPRINGS CO 80912		ABERDEEN PROVING GROUND	
		ABERDEEN MD 21005-5066	
ATTN TECHNICAL LIBRARY	1	ATTN CODE TNC (BLACKLEDGE)	1
ADVANCED RESEARCH PROJECTS AGENCY		RICH MATLOCK	1
DEPARTMENT OF DEFENSE		DIRECTOR	
WASHINGTON DC 20305		INTERCEPTOR TECHNOLOGY	
ATTN B BLAKE (BLD 146)	1	BALLISTIC MISSILE DEFENSE OFFICE	
J JENKINS (BLD 146)	1	THE PENTAGON	
TECHNICAL LIBRARY	1	WASHINGTON DC 20350	
COMMANDING OFFICER		ATTN SFAE SD ASP	1
AFSC		SFAE SD HED	1
2210 8TH STREET		DEPUTY COMMANDER	
WRIGHT PATTERSON AFB OH 45433		US ARMY STRATEGIC DEFENSE COMMAND	
ATTN EDWARD JENKINS	1	P O BOX 1500	
NAIC TANW		HUNTSVILLE AL 35807-3801	
HQ NAIC TANW		ATTN D WASHINGTON	1
4115 HEBBLE CREEK ROAD SUITE 28		W WALKER	1
WPAFB OH 45433-5623		R KRETZSCHMAR	1
ATTN J USSELTON	1	D FERGUSON JR	1
W B BAKER JR	1	COMMAND GENERAL	
TECHNICAL LIBRARY	1	US ARMY MISSILE COMMAND	
ARNOLD ENGINEERING DEVELOPMENT		AMSMI RD SS AT	
CENTER USAF		REDSTONE ARSENAL AL 35898-5252	
TULLAHOMA TN 37389		DEFENSE TECHNICAL INFORMATION	
ATTN H HUDGINS	1	CENTER	
G FRIEDMAN	1	8725 JOHN J KINGMAN ROAD	
TECHNICAL LIBRARY	1	SUITE 0944	
COMMANDING GENERAL		FORT BELVOIR VA 22060-6218	2
ARRADCOM PICATINNY ARSENAL		DIRECTOR	
DOVER NJ 07801		DEFENSE PRINTING SERVICE	
		BLDG 176 WASHINGTON NAVY YARD	
		901 M ST E	
		WASHINGTON DC 20474-5087	

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN CODE A76 TECHNICAL LIBRARY COMMANDING OFFICER CSSDD NSWC 6703 W HIGHWAY 98 PANAMA CITY FL 32407-7001	1	ATTN DR ALAN NICHOLSON MSC 5B DEFENSE INTELLIGENCE AGENCY MISSILE AND SPACE INTELLIGENCE CTR REDSTONE ARSENAL AL 35898-5500	1
ATTN DR P WEINACHT AERODYNAMICS BRANCH PROPULSION AND FLIGHT DIV WTD AMSRL WT PB US ARMY RESEARCH LAB ABERDEEN PROVING GROUND MD 21005-5066	1	ATTN EDWARD HERBERT US ARMY MISSILE COMMAND AMSMI RD MG GA BLDG 5400 ROOM 250 REDSTONE ARSENAL AL 35898	1
ATTN GREGG ABATE US AIR FORCE WRIGHT LABORATORY WL MNAA 101 W EGLIN BLVD STE 219 EGLIN AFB FL 32542-5000	1	ATTN PAUL KOLODZIEJ NASA AMES RESEARCH CENTER MS 234 1 MOFFETT FIELD CA 94035	1
ATTN JOHN GRAU US ARMY ARDEC COMMANDER US ARMY ARDEC AMSTA AR AET A BLDG 3342 PICATINNY ARSENAL NJ 07806-5000	1	ATTN LCDR T HARTLINE USNR R NR ONI 2109 NAVAL RESERVE UNIT 112 CRESTVIEW CIRCLE MADISON AL 35758	1
ATTN FRANK MACDONALD NAWC CHINA LAKE COMMANDER CODE 473 20D NAVAIRWARCENNSDNDIV CHINA LAKE CA 93555	1	ATTN CODE 4732HOD DAVID HALL PROPULSION PERFORMANCE OFFICE NAVAL AIR WARFARE CTR WEAPONS DIV 1 ADMINISTRATIVE CIR CHINA LAKE CA 93555-6001	1
ATTN MARK LAMBERT NAWC CODE 4732HOD CHINA LAKE CA 93555	1	ATTN DONALD SHEREDA WL FIMA BLDG 450 2645 FIFTH ST STE 30 WRIGHT PATTERSON AFB OH 45433-7936	1
ATTN MICHAEL MUSACHIO DIRECTOR OFFICE OF NAVAL INTELLIGENCE 4251 SUTLAND ROAD (ONI 2321) WASHINGTON DC 20395	1	ATTN ROBERT VAN DYKEN 473110D COMMANDER NAWC CHINA LAKE CA 93555	1
		BMDO AQS 1725 JEFFERSON DAVIS HWY STE 809 ARLINGTON VA 22202	1

**DISTRIBUTION (Continued)**

	<u>Copies</u>		<u>Copies</u>
ATTN JEFFREY RANDORF US ARMY SPACE AND STRATEGIC DEFENSE COMMAND P O BOX 1500 CSSD-BC-SS 106 WYNN DRIVE HUNTSVILLE AL 35807-3801	1	ATTN TECHNICAL LIBRARY NASA WASHINGTON DC 20546	1
ATTN TECHNICAL LIBRARY NEILSON ENGINEERING AND RESEARCH INC 526 CLYDE AVENUE MOUNTAIN VIEW CA 95043	1	ATTN B HENDERSON D MILLER J ALLEN F WILCOX TECHNICAL LIBRARY NASA LANGLEY RESEARCH CENTER HAMPTON VA 23365	1 1 1 1 2
ATTN TECHNICAL LIBRARY MCDONNELL DOUGLAS ASTRONAUTICS CO (WEST) 5301 BOLSA AVENUE HUNTINGTON BEACH CA 92647	1	ATTN D G MILLER (L 219) TECHNICAL LIBRARY LAWRENCE LIVERMORE NATIONAL LABORATORY EARTH SCIENCES DIVISION UNIVERSITY OF CALIFORNIA P O BOX 808 LIVERMORE CA 94551	1 1
ATTN TECHNICAL LIBRARY LOCKHEED MISSILES AND SPACE CO INC PO BOX 1103 HUNTSVILLE AL 35807	1	ATTN W RUTLEDGE (1635) R LAFARGE R EISLER TECHNICAL LIBRARY SANDIA NATIONAL LABORATORY P O BOX 5800 ALBUQUERQUE NM 87185-5800	1 1 1 1
<b>NON-DOD ACTIVITIES (CONUS)</b>		ATTN WALT GUTIERREZ SANDIA NATIONAL LABORATORIES MAIL STOP 0825 P O BOX 5800 ALBUQUERQUE NM 87185-0825	1
NICHOLS RESEARCH CORPORATION MS 912 P O BOX 400002 4040 S MEMORIAL PKWY HUNTSVILLE AL 35815-1502	1	ATTN ASSISTANT DEFENSE COOPERATION ATTACHE EMBASSY OF SPAIN WASHINGTON DC 20016	1
THE CNA CORPORATION P O BOX 16268 ALEXANDRIA VA 22302-0268	1	DE/AVT DEFENSE EQUIPMENT STAFF BRITISH EMBASSY 3100 MASSACHUSETTS AVE NW WASHINGTON DC 20008-3688	1
GIDEP OPERATIONS OFFICE CORONA CA 91720	1		
ATTN TECHNICAL LIBRARY NASA AMES RESEARCH CENTER MOFFETT CA 94035-1099	1		
ATTN C SCOTT D CURRY NASA JOHNSON SPACE CENTER HOUSTON TX 77058	1 1		

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN ASO LO IS ISRAEL AIR FORCE LIAISON OFFICER 700 ROBBINS AVE PHILADELPHIA PA 19111	1	ATTN ROBERT ENGLAR GEORGIA TECH RESEARCH INSTITUTE AEROSPACE SCIENCE AND TECHNOLOGY LAB ATLANTA GA 30332	1
ATTN GERMAN MILITARY REP US OA GMR TRAFFIC AND TRANSPORTATION DIVISION 10 SERVICES ROAD DULLES INTERNATIONAL AP WASHINGTON DC 20041	1	ATTN E LUCERO D FROSTBUTTER L PERINI TECHNICAL LIBRARY APPLIED PHYSICS LABORATORY JOHNS HOPKINS UNIVERSITY JOHNS HOPKINS ROAD LAUREL MD 20723-6099	1 1 1 1
ATTN PROF F R DEJARNETTE NORTH CAROLINA STATE UNIVERSITY DEPT OF MECHANICAL AND AEROSPACE ENGINEERING BOX 7921 RALEIGH NC 27695	1	ATTN B BROOKS R STANCIL R ELKINS LORAL VOUGHT SYSTEMS P O BOX 650003 M S EM 55 DALLAS TX 75265-0003	1 1 1
ATTN PROF J A SCHETZ VIRGINIA POLYTECHNIC AND STATE UNIVERSITY DEPT OF AEROSPACE ENGINEERING BLACKSBURG VA 24060	1	ATTN PROF J D ANDERSON DEPT OF AEROSPACE ENGINEERING UNIVERSITY OF MARYLAND COLLEGE PARK MD 20742	1
ATTN J M WU C BALASUBRAMAYAN TECHNICAL LIBRARY THE UNIVERSITY OF TENNESSEE SPACE INSTITUTE TULLAHOMA TN 37388	1 1 1	ATTN TECHNICAL LIBRARY HUGHES MISSILE SYSTEMS COMPANY P O BOX 11337 BLDG 802 MS A1 OLD NOGALES HWY TUCSON AZ 83734-1337	1
ATTN R NELSON TECHNICAL LIBRARY UNIVERSITY OF NOTRE DAME DEPT OF AEROSPACE AND MECHANICAL ENGINEERING BOX 537 NOTRE DAME IN 46556	1 1	ATTN M DILLENUS NIELSEN ENGINEERING AND RESEARCH INC 526 CLYDE AVE MOUNTAIN VIEW CA 95043	1
ATTN PROF F NELSON DEPT OF MECH AND AERO ENG UNIVERSITY OF MISSOURI ROLLA ROLLA MO 65401	1	ATTN J XERIKOS N CAMPBELL TECHNICAL LIBRARY MCDONNELL DOUGLAS ASTRONAUTICS CO (WEST) 5301 BOLSA AVE HUNTINGTON BEACH CA 92647	1 1 1

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN J WILLIAMS	1	ATTN DR G S SCHMIDT	1
S VUKELICH	1	LORAL DEFENSE SYSTEMS	
J FIVEL	1	1210 MASSILLON ROAD	
R GERBSCH (CODE 1111041)	1	AKRON OH 44315-0001	
TECHNICAL LIBRARY	1		
MCDONNELL DOUGLAS		ATTN W NORDGREN 721	1
ASTRONAUTICS CO (EAST)		GOULD INC OSD	
BOX 516		18901 EUCLID AVE	
ST LOUIS MO 63166-0516		CLEVELAND OH 44117	
ATTN TECHNICAL LIBRARY	1	ATTN TECH LIBRARY	1
UNITED TECHNOLOGIES		AEROJET ELECTRONIC SYSTEMS	
NORDEN SYSTEMS		P O BOX 296 III	
NORWALK CT 06856		AZUSA CA 91702	
ATTN T LUNDY	1	ATTN P REDING	1
D ANDREWS	1	G CHRUSCIEL	1
TECHNICAL LIBRARY	1	TECHNICAL LIBRARY	1
LOCKHEED MISSILES AND SPACE CO INC		LOCKHEED MISSILES AND SPACE CO INC	
P O BOX 1103		P O BOX 3504	
HUNTSVILLE AL 35807		SUNNYVALE CA 94088	
ATTN W CHRISTENSON	1	ATTN K C LEE	1
D WARNER	1	AEROTHERM CORP	
ALLIANT TECHSYSTEMS INC		580 CLYDE AVE	
600 SECOND ST NE		MOUNTAIN VIEW CA 94043	
HOPKINS MN 55343			
ATTN TECHNICAL LIBRARY	1	ATTN TECH LIBRARY	1
B SALEMI	1	FMC NAVAL SYSTEMS DIV	
J BOUDREAU	1	4800 E RIVER ROAD	
RAYTHEON COMPANY		MINNEAPOLIS MN 55421-1402	
MISSILE SYSTEMS DIVISION			
P O BOX 1201		ATTN DORIA GLADSTONE	1
TEWKSBURY MA 01876-0901		BATTELLE MEMORIAL INSTITUTE	
		COLUMBUS DIVISION	
		505 KING AVE	
ATTN LLOYD PRATT	1	COLUMBUS OH 43201-2693	
AEROJET TACTICAL SYSTEMS CO			
P O BOX 13400		ATTN JAMES SORENSON	1
SACRAMENTO CA 95813		VINCENT ALLEN	1
		ORBITAL SCIENCES	
ATTN JOSEPH ANDRZEJEWSKI	1	3380 SOUTH PRICE ROAD	
MEVATEC CORP		CHANDLER AZ 85248	
1525 PERIMETER PARKWAY			
SUITE 500			
HUNTSVILLE AL 35806			

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN J FORKOIS KAMAN SCIENCES CORP 1500 GARDEN OF THE GODS ROAD P O BOX 7463 COLORADO SPRINGS CO 80933	1	ATTN H A MCELROY GENERAL DEFENSE CORP P O BOX 127 RED LION PA 17356	1
ATTN RON EFROMSON MIT LINCOLN LABORATORY 244 WOOD STREET LEXINGTON MA 02173-0073	1	ATTN ENGINEERING LIBRARY ARMAMENT SYSTEMS DEPT GENERAL ELECTRIC CO BURLINGTON VT 05401	1
ATTN DJ GIESE MAIL STOP 4C 61 BOEING DEFENSE AND SPACE GROUP PO BOX 3999 SEATTLE WA 98124-2499	1	ATTN TECHNICAL LIBRARY OAYNE AERONAUTICAL 2701 HARBOR DRIVE SAN DIEGO CA 92138	1
ATTN BRIAN WALKUP ALLEGHENY BALLISTICS LAB 210 STATE ROUTE 956 WV01-13 ROCKET CENTER WV 26726-3548	1	ATTN BRIAN EST BOEING ST LOUIS P O BOX 516 ST LOUIS MO 63166-0516	1
ATTN DR T LIN TRW ELECTRONICS AND DEFENSE SECTOR BLDG 527/RM 706 P O BOX 1310 SAN BERNADINO CA 92402	1	ATTN WILLIAM FACINELLI ALLIED SIGNAL P O BOX 22200 MS 1207 3B TEMPE AZ 85285	1
ATTN G VINCENT SPARTA INC 4901 CORPORATE DR HUNTSVILLE AL 35805	1	ATTN DR T P SHIVANANDA TRW BMD P O BOX 1310 SAN BERNADINO CA 92402-1313	1
ATTN D P FORSMO TECHNICAL LIBRARY RAYTHEON COMPANY MISSILE SYSTEMS DIVISION HARTWELL RD BEDFORD MA 01730-2498	1 1	ATTN T R PEPITONE AEROSPACE TECHNOLOGY INC P O BOX 1809 DAHLGREN VA 22448	1
ATTN M S MILLER N R WALKER DYNETICS INC P O DRAWER B HUNTSVILLE AL 35814-5050	1 1	ATTN ERIC MOORE MAIL STOP MER 24 1281 LOCKHEED SANDERS P O BOX 868 NASHUA NH 03061	1
		ATTN DR BRIAN LANDRUM RI BLDG E33 PROPULSION RESEARCH CENTER UNIVERSITY OF ALABAMA HUNTSVILLE AL 35899	1

**DISTRIBUTION (Continued)**

	<u>Copies</u>		<u>Copies</u>
ATTN BRUCE NORTON MAIL STOP BL 1 RAYTHEON 100 VANCE TANK RD BRISTOL TN 37620	1	ATTN THOMAS FARISS LOCKHEED SANDERS P O BOX 868 MER24 1206 NASHUA NH 03061-0868	1
ARROW TECH ASSOCIATES INC 1233 SHELburne ROAD D8 SO BURLINGTON VT 05403		ATTN JEFFREY HUTH KAMAN SCIENCES CORPORATION 2560 HUNTINGTON AVE ALEXANDRIA VA 22303	1
ATTN JUAN AMENABAR SAIC 4001 NORTH FAIRFAX DRIVE STE 800 ARLINGTON VA 22209	1	ATTN WILLIAM JOLLY KAMAN SCIENCES 600 BLVD SOUTH SUITE 208 HUNTSVILLE AL 35802	1
ATTN TECHNICAL LIBRARY TELEDYNE RYAN AERONAUTICAL 2701 HARBOR DRIVE SAN DIEGO CA 92138	1	ATTN STEPHEN MALLETTE KBM ENTERPRISES 15980 CHANEY THOMPSON RD HUNTSVILLE AL 35803	1
ATTN DR KIRIT PATEL SVERDRUP TECHNOLOGY INC TEAS GROUP BLDG 260 P O BOX 1935 EGLIN AFB FL 32542	1	ATTN DONALD MOORE NICHOLS RESEARCH CORPORATION 4040 SOUTH MEMORIAL PARKWAY P O BOX 400002 MS 920C HUNTSVILLE AL 35815-1502	1
ATTN FRANK LANGHAM MICRO CRAFT TECHNOLOGY 740 4TH ST MS 6001 ARNOLD AFB TN 37389	1	ATTN NANCY SWINFORD LOCKHEED MISSILES & SPACE CO P O BOX 3504 ORG E5-40 BLDG 1575E SUNNYVALE CA 94088-3504	1
ATTN LAURA AYERS DELTA RESEARCH INC 315 WYNN DRIVE SUITE 1 HUNTSVILLE AL 35805	1	ATTN DAVID RESSLER TRW BALLISTIC MISSILES DIV MS 953 2420 P O BOX 1310 SAN BERNARDINO CA 92402	1
ATTN BRIAN BENNETT MCDONNELL DOUGLAS MC 064 2905 P O BOX 516 ST LOUIS MO 63166-0516	1	ATTN MARK SWENSON ALLIANT TECHSYSTEMS MN11 262B 600 SECOND STREET NE HOPKINS MN 55343	1

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN LEROY M HAIR COLEMAN RESEARCH CORP 6820 MOQUIN DRIVE HUNTSVILLE AL 35806	1	ATTN EUGENE HART SYSTEM PLANNING CORP 1000 WILSON BLVD ARLINGTON VA 22209	1
ATTN SCOTT ALLEN ALLEN AERO RESEARCH 431 E SUNNY HILLS RD FULLERTON CA 92635	1	ATTN ELAINE POLHEMUS ROCKWELL AUTONETICS & MISSILE SYSTEMS DIVISION D611 DL23 1800 SATELLITE BLVD DULUTH GA 30136	1
ATTN DARRYL HALL SAIC 997 OLD EAGLE SCHOOL RD SUITE 215 WAYNE PA 19087-1803	1	ATTN MICHAEL GLENN TASC 1992 LEWIS TURNER BLVD FT WALTON BEACH FL 32547	1
ATTN PETER ALEXANDER MCDONNELL DOUGLAS AEROSPACE 689 DISCOVERY DRIVE MS 11A1 HUNTSVILLE AL 35806	1	ADAPTIVE RESEARCH 4960 CORPORATE DRIVE SUITE 100 A HUNTSVILLE AL 35805-6229	1
ATTN SAMUEL HICKS III TEXAS INSTRUMENTS 6600 CHASE OAKS BLVD MS 8490 PLANO TX 75086	1	ATTN STEVEN MARTIN SYSTEMS ENGINEERING GROUP INC 9841 BROKEN LAND PARKWAY SUITE 214 COLUMBIA MD 21046-1120	1
ATTN BARRY LINDBLOM ALLIANT DEFENSE ELECTRONICS SYSTEMS INC P O BOX 4648 CLEARWATER FL 34618	1	ATTN C W GIBKE LOCKHEED MARTIN VOUGHT SYSTEMS MS SP 72 P O BOX 650003 DALLAS TX 75265-0003	1
ATTN DR SHIN CHEN THE AEROSPACE CORP M4 967 P O BOX 92957 LOS ANGELES CA 90009	1	ATTN CHRIS HUGHES EDO GOVERNMENT SYSTEMS DIV 14 04 111TH ST COLLEGE POINT NY 11356	1
ATTN ROBERT ACEBAL SAIC 1225 JOHNSON FERRY RD SUITE 100 MARIETTA GA 30068	1	ATTN DANIEL LESIEUTRE NIELSEN ENGINEERING & RES INC 526 CLYDE AVENUE MOUNTAIN VIEW CA 94043-2212	1

**DISTRIBUTION (Continued)**

	<u>Copies</u>		<u>Copies</u>
ATTN CARL HILL	1	ATTN ROBERT BRAENDLEIU	1
FRANCIS PRIOLO	1	KAISER MARQUARDT	
STANDARD MISSILE COMPANY LLC		16555 SATICOY ST	
1505 FARM CREDIT DRIVE		VAN NUYS CA 91406-1739	
SUITE 600			
MCLEAN VA 22102		ATTN THOMAS LOPEZ	1
		COLEMAN RESEARCH CORP	
ATTN JENNIE FOX	1	990 EXPLORER BLVD	
LOCKHEED MARTIN VOUGHT SYSTEMS		HUNTSVILLE AL 35806	
P O BOX 650003			
MS EM 55		ATTN LAWRENCE FINK	1
DALLAS TX 75265-0003		BOEING DEFENSE AND SPACE GROUP	
		P O BOX 3999 MS 82-23	
ATTN JOHN BURKHALTER	1	SEATTLE WA 98124	
AUBURN UNIVERSITY			
211 AEROSPACE ENGR BLDG		ATTN ROY KLINE	1
AUBURN UNIVERSITY AL 36849		KLINE ENGINEERING CO INC	
		27 FREDON GREENDELL RD	
ATTN DR MAX PLATZER	1	NEWTON NJ 07860-5213	
NAVAL POSTGRADUATE SCHOOL			
DEPT OF AERONAUTICS &		ATTN THOMAS KLAUSE	1
ASTRONAUTICS		TRW	
CODE AA PL		P O BOX 80810	
MONTEREY CA 93943		ALBUQUERQUE NM 87198	
ATTN MIKE DANVELO	1	ATTN DAN PLATUS	1
MIT LINCOLN LABORATORY		THE AEROSPACE CORPORATION	
1745 JEFFERSON DAVIS HWY 1100		P O BOX 92957	
ARLINGTON VA 22202		LOS ANGELES CA 90009	
ATTN RICHARD HAMMER	1	ATTN DR REX CHAMBERLAIN	1
JOHNS HOPKINS APPLIED PHYSICS LAB		TETRA RESEARCH CORPORATION	
JOHNS HOPKINS ROAD		2610 SPICEWOOD TR	
LAUREL MD 20723-6099		HUNTSVILLE AL 35811-2604	
ATTN MAURICE TUCKER	1	ATTN DR DANNY LIU	1
BATTELLE HUNTSVILLE OPERATIONS		ZONA TECHNOLOGY INC	
7501 S MEMORIAL PKWY STE 101		2651 W GUADALUPE RD SUITE B 228	
HUNTSVILLE AL 35802		MESA AZ 85202	
ATTN STEVE MULLINS	1	ATTN PERRY PETERSEN	1
SIMULATION AND ENGINEERING CO INC		NORTHROP GRUMMAN CORP	
8840 HWY 20 STE 200 N		DEPT 9B51 MAIL ZONE XA	
MADISON AL 35758		8900 EAST WASHINGTON BLVD	
		PICO RIVERA CA 90660-3783	

**DISTRIBUTION (Continued)**

	<u>Copies</u>		<u>Copies</u>
ATTN DR JAMES HAUSER AERO SPECTRA INC 2850 KENYON CIRCLE P O BOX 3006 BOULDER CO 80307	1	ATTN JAMES JONES SPARTA INC 1901 N FORT MYER DR SUITE 600 ARLINGTON VA 22209	1
ATTN DARRELL AUSERMAN TRW SPACE AND DEFENSE ONE SPACE PARK MAIL STATION R1-1062 REDONDO BEACH CA 90278-1071	1	ATTN SCOTT HOUSER PHOENIX INTEGRATION 1872 PRATT DRIVE SUITE 1835 BLACKSBURG VA 24060	1
ATTN JAY EBERSOHL ADVATECH PACIFIC INC 2015 PARK AVENUE SUITE 8 REDLANDS CA 92373	1	ATTN S ROM MURTY TELEDYNE BROWN ENGINEERING MS 200 300 SPARKMAN DRIVE HUNTSVILLE AL 35807	1
ATTN EDWARD RAWLINSON SY TECHNOLOGY INC 4900 UNIVERSITY SQUARE SUITE 8 HUNTSVILLE AL 35816	1	ATTN STUART COULTER SVERDRUP TECHNOLOGY 670 2ND ST MS4001 ARNOLD AIR FORCE BASE TULLAHOA TN 37389-4001	1
ATTN LAYNE COOK UNIVERSAL SPACE LINES 8620 WOLFF CT SUITE 110 WESTMINSTER CO 80030	1	ATTN DR RICHARD HOWARD NAVAL POSTGRADUATE SCHOOL DEPT OF AERONAUTICS AND ASTRONAUTICS CODE AA HO NPS MONTEREY CA 93943	1
ATTN PAUL WILDE ACTA INC 2790 SKYPARK DR SUITE 310 TORRANCE CA 90505-5345	1	ATTN J BRENT RUMINE MIT LINCOLN LABORATORY 244 WOOD STREET BUILDING S ROOM 52-327 LEXINGTON MA 02173-9185	1
ATTN DR MICHAEL HOLDEN CALSPAN UB RESEARCH CENTER P O BOX 400 BUFFALO NY 14225	1	<b>NON-DOD ACTIVITIES (EX-CONUS)</b>	
ATTN RICHARD GRABOW SPACE VECTOR CORP 17330 BROOKHURST ST SUITE 150 FOUNTAIN VALLEY CA 92708	1	ATTN LOUIS CHAN INSTITUTE FOR AEROSPACE RESEARCH NATIONAL RESEARCH COUNCIL MONTREAL RD OTTAWA ONTARIO CANADA K1A0R6	1
ATTN BRENT APPLEBY DRAPER LABORATORY 555 TECHNOLOGY SQ MS77 CAMBRIDGE MA 02139	1		

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN H B ASLUND SAAB MILITARY AIRCRAFT 581 88 LINKÖPING SWEDEN	1	ATTN R CAYZAC GIAT INDUSTRIES 7 ROUTE DE GUERCY 18023 BOURGES CEDEX FRANCE	1
ATTN A BOOTH BRITISH AEROSPACE DEFENCE LTD MILITARY AIRCRAFT DIVISION WARTON AERODROME WARTON PRESTON LANCASHIRE PR4 1AX UNITED KINGDOM	1	ATTN MAJ F DE COCK ECOLE ROYALE MILITAIRE 30 AV DE LA RENAISSANCE 1040 BRUXELLES BELGIUM	1
ATTN JEKERÖT BOFORS MISSILES 691 80 KARLSKOGA SWEDEN	1	ATTN P LEZEAUD DASSAULT AVIATION 78 QUAI MARCEL DASSAULT 92214 SAINT CLOUD FRANCE	1
ATTN CH FRANSSON NATIONAL DEFENCE RESEARCH ESTABLISHMENT DEPT OF WEAPON SYSTEMS EFFECTS AND PROTECTION KARLAVAGEN 106B 172 90 SUNDBYBERG SWEDEN	1	ATTN J LINDHOUT N L R ANTHONY FOKKERWEG 2 1059 CM AMSTERDAM THE NETHERLANDS	1
ATTN M HARPER BOURNE DEFENCE RESEARCH AGENCY Q134 BUILDING RAE FARNBOROUGH HAMPSHIRE QU14 6TD UNITED KINGDOM	1	ATTN A MICKELLIDES GEC MARCONI DEFENCE SYSTEMS LTD THE GROVE WARREN LANE STANMORE MIDDLESEX UNITED KINGDOM	1
ATTN A H HASSELROT FFA P O BOX 11021 161 11 BROMMA SWEDEN	1	ATTN K MOELLER BODENSEEWERK GERAETETECHNIK GMBH POSTFACH 10 11 55 88641 UBERLINGEN GERMANY	1
ATTN B JONSSON DEFENCE MATERIAL ADMINISTRATION MISSILE TECHNOLOGY DIVISION 115 88 STOCKHOLM SWEDEN	1	ATTN G MOSS ROYAL MILITARY COLLEGE AEROMECHANICAL SYSTEMS GROUP SHRIVENHAM SWINDON WILTS SN6 8LA UNITED KINGDOM	1

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN RIBADEAU DUMAS MATRA DEFENSE 37 AV LOUIS BREGUET BP 1 78146 VELIZY VILLACOUBLAY CEDEX FRANCE	1	ATTN DR R G LACAU AEROSPATIALE MISSILE DEPT E/ECN CENTRE DES GATINES 91370 VERRIERE LE BUISSON FRANCE	1
ATTN R ROGERS DEFENCE RESEARCH AGENCY BLDG 37 TUNNEL SITE CLAPHAM BEDS MK 41 6AE UNITED KINGDOM	1	ATTN J M CHARBONNIER VON KARMAN INSTITUTE 72 CHAUSSEE DE WATERLOO 1640 RHODE SAINT GENESE BELGIUM	1
ATTN S SMITH DEFENCE RESEARCH AGENCY Q134 BUILDING RAE FARNBOROUGH HAMPSHIRE QU14 6TD UNITED KINGDOM	1	ATTN P CHAMPIGNY DIRECTION DE L AERONAUTIQUE ONERA 29 AV DE LA DIVISION LECLERC 92320 CHATILLON SOUS BAGNEUX CEDEX FRANCE	1
ATTN J SOWA SAAB MISSILES AB 581 88 LINKOPING SWEDEN	1	ATTN DR P HENNIG DEUTSCHE AEROSPACE (DASA) VAS 414 ABWEHR AND SCHUTZ POSTFACH 801149 8000 MUENCHEN 80 GERMANY	1
ATTN D SPARROW HUNTING ENGINEERING LTD REDDINGS WOOD AMPTHILL BEDFORDSHIRE MK452HD UNITED KINGDOM	1	ATTN H G KNOCHE DR GREGORIOU MESSERSCHMIDT BOLKOW BLOHM GMBH UNTERNEHMENSBEREICH APPARATAE MUNCHEN 80 POSTFACH 801149 BAYERN GERMANY	1 1
ATTN P STUDER DEFENCE TECHNOLOGY AND PROCUREMENT AGENCY SYSTEMS ANALYSIS AND INFORMATION SYSTEMS DIVISION PAPIERMUEHLESTRASSE 25 3003 BERNE SWITZERLAND	1	ATTN DR S J YOON AGENCY FOR DEFENSE DEVELOPMENT AERODYNAMICS DIVISION (4-3-1) P O BOX 35-4 YUSEONG TAEJON KOREA	1

## DISTRIBUTION (Continued)

	<u>Copies</u>		<u>Copies</u>
ATTN PETER CAAP	1	G30	1
HD FLIGHT SYS DEPT		G305	1
FAA AERONAUTICAL RESEARCH INST		G32 (DAY)	1
OF SWEDEN		G33 (MELTON)	1
BOX 11021		G33 (RINALDI)	1
BROMMA SWEDEN 16111		G50	1
		G50 (SOLOMON)	1
ATTN DAVE BROWN	1	G60	1
WEAPON SYSTEMS DIVISION		G70	1
AERONAUTICAL AND MARITIME		G72	1
RESEARCH LABORATORY		G72 (ALEXOPOULOS)	1
P O BOX 1500 SALISBURY		G72 (CHEPREN)	1
SOUTH AUSTRALIA 5108		G72 (JONES)	1
		G72 (ROBINSON)	5
INTERNAL		G72 (MCINVILLE)	1
		K	1
B	1	K40	1
B04	1	K44 (ICHNIOWSKI)	1
B04 (ZIEN)	1	K44 (JOHNSON)	1
B05 (GRAFF)	1	N	1
B05 (STATON)	1	N93 (NEGRON)	1
B10	1	T	1
B10 (HSIEH)	1	T406	1
B51 (ARMISTEAD)	1		
B60 (TECHNICAL LIBRARY)	3		
B60A (GRAY)	1		
C	1		
D	1		
G	1		
G02	1		
G04	5		
G20	1		
G205	1		
G205 (ELLIOTT)	1		
G205 (LONG)	1		
G23	1		
G23 (BIBEL)	1		
G23 (CHADWICK)	1		
G23 (COOK)	1		
G23 (HANGER)	1		
G23 (HARDY)	1		
G23 (HOCK)	1		
G23 (HYMER)	1		
G23 (MALYEVAC)	1		
G23 (OHLMEYER)	1		
G23 (ROWLES)	1		
G23 (WEISEL)	1		